华兴数控
WASHING

31DT LCD Screen Lathe Machine Tool
Numerical Control System

Operation & Programming Guide

NANJING WASHING CNC DEVICE CO.,LTD

MENU

Chapter One 31DT System Overview 1
1.1 31DT system indicator 1
1.1.1 Main specifications 1
1.1.2 System resource 1
1.2 Coordinate system provision 1
1.2.1 Movement principles which are relative to stationary work piece 1
1.2.2 Standard coordinate system provision 2
1.2.3 Provision on machine tool moving component direction 2
1.2.4 Machine tool reference point 2
1.2.5 Work piece coordinate system 2
1.2.6 Cutting tool compensation principle when changing cutting tool 3
Chapter Two Programming 4
2.1 Block format 4
2.1.1 Parameter 5
2.2 Preparative function (G function) 6
2.2.1 G00－Fast locating 7
2.2.2 G01－Line interpolation 7
2.2.3 G02—Interpolation of clockwise arc 8
2.2.4 G03—Inverse arc interpolation 9
2.2.5 G04—Time delay 9
2.2.6 G09－Feed angle-specified stop 9
2.2.7 G20－Sub-program call 9
2.2.8 G22－Sub-program definition 10
2.2.9 G24—Completion and returning of sub-program 10
2.2.10 G25－Jump processing. 11
2.2.11 G26－Transition processing (sub-program call in the program) 11
2.2.12 G27—Endless cycle 11
2.2.13 G30—Magnification and minification cancel 12
2.2.14 G31－Magnification or minification 12
2.2.15 G47－Short line speed auto-transition 12
2.2.16 G48—Cancel G47. 12
2.2.17 G53—Cancel zero bias 12
2.2.18 G54－Absolute zero bias 12
2.2.19 G55－Increment zero bias 13
2.2.20 G56－Current point bias 13
2.2.21 G71－Internal (external) diameter cutting compound cycle 14
2.2.22 G72—End surface cutting compound cycle 15
2.2.23 G73—Sealed contour compound cycle 16
2.2.24 G74—Returning to reference point (mechanical origin) 19

2.2.25 G75－Returning to presetting cutter point from reference point.....19
2.2.26 G76—Returning to processing start point from current position (feed point)..19
2.2.27 G77—Recovering current coordinate system.....20
2.2.28 G81－Excircle (inner circle) fixed cycle.....20
2.2.29 G82－End surface fixed cycle.....22
2.2.30 G85－Threading cycle.....24
2.2.31 G86－Metric thread cycle.....24
2.2.32 G87－English system thread cycle. 27
2.2.33 G90－Programming with absolute value mode 27
2.2.34 G91－Programming with incremental mode 27
2.2.35 G92－Setting work piece coordinate system 27
2.2.36 G96－Constant linear speed cutting 28
2.2.37 G97－Cancel constant linear speed cutting.....28
2.2.38 G98－Cancel feed of each rotation.....28
2.2.39 G99－Setting feed of each rotation.....28
2.3 Subsidiary function (M function) 28
2.3.1 M00－Program pause.....29
2.3.2 M01－Condition pause.....29
2.3.3 M02－Program completion.....29
2.3.4 M03－Spindle positive rotation 30
2.3.5 M04－Spindle reversion.....30
2.3.6 M05－Spindle stop.....30
2.3.7 M08—Open cooling fluid 31
2.3.8 M09－Close cooling fluid 31
2.3.9 M10－Work piece clamping.....31
2.3.10 M11—Work piece loosing.....31
2.3.11 M20－Open specified relay 32
2.3.12 M21－Close specified relay.....32
2.3.13 M24－Setting cutting tool compensation number 32
2.3.14 M30－Returning to program header 32
2.3.15 M71~M85－M function pulse output.....32
2.3.16 Output interface, programming and relay number comparison table.....32
2.4 F、S、T functions 33
2.4.1 F－Feed function.....33
2.4.2 S－Spindle speed control.....33
2.4.3 T－Cutting tool function 34
2.4.4 Axis optional function.....38
2.4.5 Coordinate modification and regulation function.....38
Chapter Three System Operation 39
3.1 Safety, protection and compensation.....39
3.1.1 Emergency stop.....39
3.1.2 Hard limit 39
3.1.3 Soft limit 40
3.1.4 Clearance compensation.....40

3.1.5 Screw thread pitch compensation....40
3.2 Operating keyboard of numerical control system....41
3.2.1 Primary function key....42
3.2.2 Edit character key....42
3.2.3 Coordinate feed and feed parameter setting of manually operating machine tool....42
3.2.4 Soft definition key F1~ F5....42
3.2.5 Others....42
3.3 Starting up....44
3.3.1 Starting up, image and design....44
3.3.2 Selection of primary function....45
3.3.3 Selection of sub-function....45
3.4 PRGRM (Program) primary function....46
3.4.1 Input principle of program name....46
3.4.2 Program edit....47
3.4.3 Copy, delete and program status....48
3.4.4 Rename, input and output function....49
3.4.5 List....50
3.5 OPERT (Process) primary function....51
3.5.1 Auto-cycle (including start processing in the position of any block number)....52
3.5.2 Manual operation of machine tool....52
3.5.3 Return to machine tool zero....53
3.5.4 Hand wheel (hand-operated impulse generator)....53
3.5.5 System status setup....54
3.5.6 MDI operating mode....54
3.6 Figure display function....54
3.6.1 Image access sequence of figure display function....55
3.6.2 Image selection of figure display function....55
Chapter Four Parameter function....57
4.1 31DT parameter system....57
4.2 Basic conception of parameter....58
4.2.1 Time constant of speed increase and decrease....58
4.2.2 Acceleration....58
4.2.3 Electronic gear ratio....58
4.2.4 Parameter password....59
4.3 System parameter (P parameter)....59
4.3.1 Meaning of P parameter....60
4.4 Digit Parameter....63
4.4.1 Access....63
4.4.2 Digit parameter setting....63
4.5 Thread pitch error compensation....67
4.5.1 The required caution problems of thread pitch error compensation....68
4.5.2 Thread pitch error compensation example....68
4.6 Spindle....69

4.7 Cutting tool parameters 69
4.8 Coordinate modification and regulation 70
4.9 Diagnosis (external input signal monitoring) 70
4.10 System zero setting 71
4.10.1 Clear memory 71
4.10.2 Format 71
4.10.3 Password setup 71
4.10.4 Backup 72
4.11 Shutdown and starting up 73
Appendix One--Error alarm 74
Appendix Three Programming Example 76

Chapter One 31DT System Overview

1.1 31DT system indicator

1.1.1 Main specifications

Pulse equivalent	**X: 0.001mm Z: 0.001mm**
Control/compounding axis number	**2／2**
Programming scope	**-9999.999~+9999.999mm**
Foward speed	**60000 mm/min (0.001 mm equivalent)**
Program capacity	**Electronic disk 128K, which can store 30 programs**
nterpolation	**Line, arc, metric system, English system straight/cone, multi-thread/single thread**

1.1.2 System resource

Display	**6″ LCD is adopted with 320X240 dot matrix**
Electronic disk	**128KB memory for saving program status and parameter**
Input signal	**24-way on-off and optical isolation**
Hand wheel interface	**1 way, 1, 10, 100 multiply factor**
Encoder interface	**1 way, four-multiple frequency processing**
Output signal	**17-way on-off.. 12-way relay power driving output and 5-way relay contact output**
	Stepping motor driving signal (CP and CW) output in X and Z directions
	One-way 8-bit analogue output, 0－5V or 0－10V
Serial port	**RS232C asynchronous serial port**

1.2 Coordinate system provision

When the elements are processed on the numerically controlled machine tool, the relative movement between cutting tool and elements must be in the specified coordinate system, and then the program can be processed according to provisions. For the convenience of describing machine tool movement when programming, simplifying the program compilation methods to guaranty the exchangeability of recorded data, and the coordinate and movement direction of numerically controlled machine tool have been standardized.. The Ministry of Engineering Industry promulgated the denomination standards of JB 3051-82 numerically controlled machine tool coordinate and movement direction in 1982, and the denomination principles and provisions are as follows:

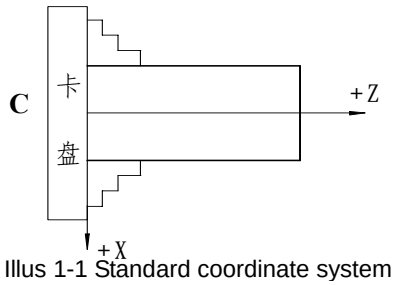
1.2.1 Movement principles which are relative to stationary work piece

This principle is for programming staves to determine machine tool operating process according to element drawings under condition that they don't know whether it's the cutting tool movement or work piece movement..

1.2.2 Standard coordinate system provision

The standard coordinate system is one rectangular coordinate system. as the figure of previous page:.

Various coordinate axes of this coordinate system parallel to primary guide tracks of machine tool.



Illus 1-1 Standard coordinate system

1.2.3 Provision on machine tool moving component direction

The movement positive direction of certain moving component of machine tool is the one to increase distance between cutting tool and work piece.

1.2.3.1 Z coordinate movement

The Z coordinate movement is provided by spindle which transmits cutting power. In the standard coordinate system, the coordinate always paralleling to spindle is provided as Z coordinate.

1.2.3.2 X coordinate movement

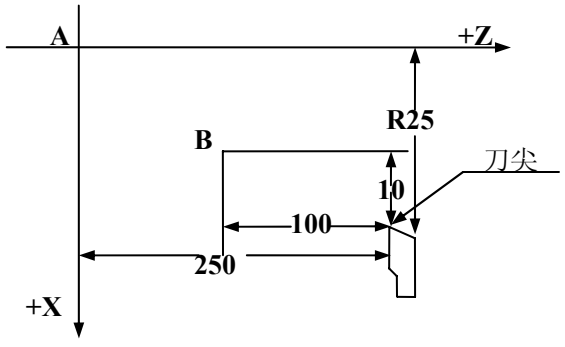
The X coordinate is horizontal, which parallels to clamping surface of work piece. The X coordinate is the movement primary coordinate in the locating planes of cutting tool or work piece.

On the lathe, the Z coordinate positive direction is that the big tool carriage moves to end bracket side along lathe bed (vertical), and X coordinate positive direction is that holder moves to handgrip direction.

1.2.4 Machine tool reference point

The machine tool reference point is also called mechanical zero, which indicates X and Z directions to move to the approaching limit position along positive direction, and induces the determined position of reference point switch in this direction. Whether one machine tool has reference point returning function depends on whether machine tool manufacturer installs reference point switch (also called mechanical origin switch).

1.2.5 Work piece coordinate system



All the coordinate values are positions of tool nose relative to coordinate origin. . If the Coordinate origin is different, even if the tool nose is at the same absolute position on the machine tool, its coordinate value is also different. . In order to guaranty uniqueness of tool nose coordinate in processing, the coordinate origin (also called zero) must be determined, while zero position is acquired by opposite calculating of tool nose position and coordinate value..

For instance: If the tool nose coordinate is supposed to be (50, 250), then the 25 mm position along X negative direction is the X coordinate origin; the 250 mm position along Z negative direction is the Z coordinate origin (see A position in the right figure)..

Note: On the lathe, the X direction coordinate (also called horizontal direction) is provided

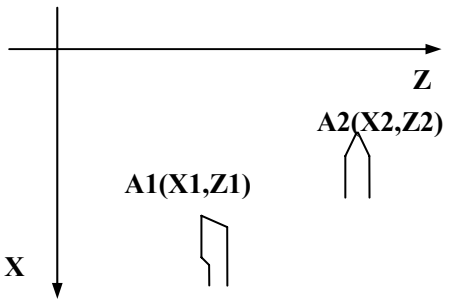
as diameter amount..

Now supposing that the tool nose position is unchanged, while the coordinate is (20,100), then zero is in the B position in the figure, which is the notion of floating zero. However, towards to one processing program, the processing can be implemented only after determining zero and optional change is prohibited (unless through instruction of coordinate transition). Once the floating zero is determined, the work piece coordinate system used in the actual processing will be composed. All tool nose movements in the program are based on this coordinate system for reference. . Seeing G92 instruction to determine coordinate zero.

1.2.6 Cutting tool compensation principle when changing cutting tool

When the comparatively complicated work piece is processed, more cutting tools are frequently required. However, the processing program is compiled according to some one tool nose of cutting tool. After changing cutting tool, the offset must occur in X and Z directions of current tool nose relative to previous one. That's to say even if big and small tool carriages don't move, the tool nose position will change after changing cutting tool, and the effect of cutting tool compensation is used to compensate this change.1

For instance: If current cutting tool is T1, its tool nose position is A1; after changing for No. 2 cutting tool (T2), the tool nose of it is in the A2 position. The tool nose coordinate is changed from A1 (X1,Z1) to A2 (X2,Z2) after changing cutting tool, and the effect of cutting tool compensation is to convert the original coordinate (X1,Z1) of tool nose coordinate value to (X2,Z2). The relative difference in X and Z directions of A1 and A2 can be measured beforehand, and this difference is the cutting tool compensation value memorized by numerical control system. In practical applications, in order to simplify this course, the numerical control system doesn't measure difference of each other among cutting tools, but adopts simpler methods to memorize cutting tool compensation value.Th That's the method of memorizing coordinate value for determination..



For instance: Contacting certain fixed point (core rod or specimen) one by one along X and Z directions for tool nose of each cutting tool, and this fixed point contacted by tool nose is regarded as standard. Owing to difference of various cutting tool lengths, the displayed coordinate point is also different contacting to fixed point. The numerical control system respectively memorizes coordinate values contacted by various cutting tool. These different coordinate values of each other actually include the length difference information between the two cutting tools. Many methods can be utilized to generate cutting tool compensation value, and 31DT adopts the method that inputting work piece size after one cut, which is equivalent to regard the presetting cutter as benchmark. It can calculate cutting tool compensation values of excircle, internal hole and so on, moreover eliminate errors brought about by process system elastic deformation.

Chapter Two Programming

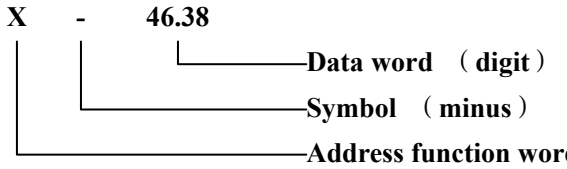
2.1 Block format

The block format means the the wirtten rules of block, it includes the function which will be implemtented by numerically controlled machine tool and parameters that are required implementing this function. One part processing program is composed of several blocks, and each block is composed of various functions; the usual function words which are used by lathe numerical control system are as follows:.

Function	Address	Scope	Meaning
Program number	P、N	01~99	Specifying program number and sub-program number
Sequence segment number	N	0000~9999	Program segment number
Preparatory function	G	00~99	Instruction movement mode
Coordinate letter	X、Z、I、K、R、L、J、D	±0.001~±9999.999	Movement instruction coordinate, circle center coordinate, thread pitch, radius and cycle number
Feed speed	F	1~60000mm／min	Feed speed instruction
Spindle function	S	0~5000RPM	Instruction of spindle rotating speed
Cutting tool function	T	1~8	Cutting tool instruction
Auxiliary function	M	0~99	Auxiliary instruction

The numerical control system doesn't require each block having these instructions above, however in every block; the instruction should be arried according to certain format.. Each function word probably has various definitions in different block definitions, refering to specific instructions..

31DT numerical control system adopts the variable block format, which means that the block length changes with word number and word length.. One block is composed of one or more program words.. Generally, the program word is composed of address word and post-address word number and symbol, for instance:



This program word format is composed of address function word as the header following a string of digits, and several words compose one block. . The words written in the previous block and unchanged in this block are also valid, which can be no longer rewritten.. In the dimension words, it can only write valid digit and not specify that every word should write the full number..

For instance: N0420 G03 X70 Z－40 10 K－20 F100

In the previous segment of program, N、G、X、Z、I、K、F are all address function words

N Block number.

G03 Preparative function, which can be written as G3.

X Z I K Coordinate address.

F Amount of feed.

"=" "-" Signifying symbol

03 70 -40 0 -20 100 are data words

In the block, the English letters signifying address function can be divided into dimension word address and non-dimension word. The dimension word address is signified with the following letters:X、Z、I、K、R、J、D, and non-dimension word address is signified with the following letters: N、S、T、G、F、M、P、l. All dimensions are signified with diameter or diameter difference in X direction.. For instance, X50 signifies that tool nose moves to ¤50 position, and 110 signifies that the diameter difference of circle center relative to arc start point is ⊿φ10..

One complete program is composed of program name, block number and corresponding symbols, please the program below.

```
N0010  G92  X50  Z100
N0020  S1200  M03
N0030  G01  X40  F300
N0040  Z90
N0050  G02  X30  Z85  I0  K-5
N0060  G01  Z60
N0070  G02  X40  Z55  I10  K0
N0080  G01  X51
N0090  G76  X  Z
N0100  M02
```

In general condition, one block is one process step of element processing, numerical control program is one block statement sequence which is stored in the memorizer.. When the elements are processed, these statements are totally read from memorizer and explained into executable data format and then executed..

The block number is used to identify every block composing program; it's composed of N with following 0000-9999. The block number must be written at the beginning of every segment, which can be generated by segment number automatic generater.. (see program edit function) In one program, the block number can adopt optionlal value in 0000-9999, however in principle, various block numbers should be arrayed from small to big according to its precedence in the program.. For the convenience of inserting new block in the required position, it's recommended not to use continual serial no. to the block when programming, and if the programming is implemented on the CNC panel, it's suggested that the block be numbered with interval of 10. In this way, it's easy to assign different block numbers when inserted into the program.. (parameter 27#)

2.1.1 Parameter

The parameters (P0-P9) can be used to substitute digit in the block.. First the assignment statement in the program can be used for parameter (P0-P9) assignment, and in the subsequent programs, the assigned parameter can be used to substitute this numerical value. When the program is automatically excuted, the parameter is changed to the lately assignment number of this variable.. If this parameter is assigned again in the program, then the new value is only

valid to the changed quotation, and the previous quotation still remains the original value..

For instance:

N0010 P2=1 P5=55 P7=200

N0020 GP2 XP5 Z100 FP7

N0030 F2=40 P5=160

N0040 XP2 ZP5

N0050 M02

When the program is executed, it's identical with:

N0020 G1 X55 Z100 F200

N0040 X40 Z160

N0050 M02

2.2 Preparative function (G function)

The preparative function is programmed with G following two digits; it's also called F instruction, which is used to define geometry and CNC operating status of track.. The functions of any numerical control device all include fundamental functions and optional functions.. The fundamental functions are requisite for system, and the optional functions are for user to select according to machine tool features and applications. It's recommended to program after understanding the machine specifications first.. The machine tool can configure control function according to numerical control system, namely the machine tool may not implement all functions of numerical control system.

The whole G functions of 31DR series numerical control system are as follows:

Modality	**G00**	**Fast locating.**
Modality	**G01**	**Line interpolation.**
Modality	**G02**	**Clockwise circular interpolation or spiral interpolation**
Modality	**G03**	**Anti clockwise circular interpolation or spiral interpolation.**
	G04	**Time delay**
	G09	**Servo angle-specified stop and positioning**
	G20	**Independent sub-program call**
	G22	**Independent of program definition**
	G24	**Independent of program definition completion, and returning to call program**
	G25	**Skip processing**
	G26	**Block calls processing (sub-program call in the program)**
	G27	**Endless loop**
Modality	**G30**	**Multiply factor cancels**
Modality	**G31**	**Multiply factor definition**
	G47	**Initiating short line smooth transition function**
	G48	**Cancel short line smooth transition function**
	G53	**Cancel zero bias**
	G54	**Absolute value zero bias**
	G55	**Relative value zero bias**
	G56	**Current point zero bias**
	G71	**Internal (external) diameter cutting compound cycle**

	G72	End surface cutting compound cycle
	G73	High speed deep hole processing cycle.
	G74	Return to reference point (mechanical origin).
	G75	Return to the presetting cutter point.
	G76	Return to the program zero from the current position.
	G78	Fine boring cycle.
	G81	Center hole drilling cycle
	G82	Center hole drilling cycle with pause
	G85	English system rigidity threading cycle
	G86	Boring cycle (auto-return)
	G87	Counter boring cycle.
Modality	G90	Absolute value mode programming.
Modality	G91	Increased value mode programming
	G92	Setting up program zero
	G96	Constant linear speed cutting
	G97	Cancel constant linear speed cutting
	G98	Cancel feed of each rotation
	G99	Setting feed of each rotation
	P=	Parameter assignment.

Note: The modality maintains valid after this G function is programmed until superseded by another modality function of the same quality..

Now, the G functions above will be elaborated.

2.2.1 G00—Fast locating

Format: G 00 X _Z _

Description: (1) All the programming axes move simultaneously at the rate which is defined by 0# parameter, and when certain axle stops after completing programming value, while other axes move on.

(2) The non-movement coordinate requires no programming..

(3) The coordinate value of target point can use absolute value, and can also use increased value. Most 5 digits are allowed before decimal, and after decimal most 3 digits are allowed, the positive number can omit "+" (this rule is suitable for all coordinate programming).

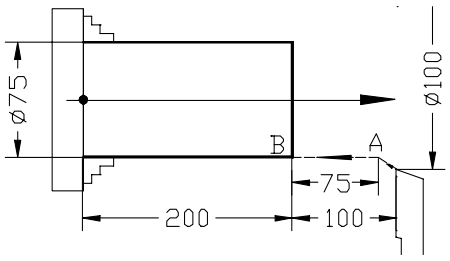


图 2-1 G00

(4) When G00 is programmed, it's also written as G0.

Example: The right figure program is as follows:

Absolute value mode programming: G00 X75 Z200.

Incremental mode programming: G91 G00 X-25 Z-100

First X and Z move fast simultaneously to A point, then Z. moves 75 fast to B point.

2.2.2 G01—Line interpolation

Format: G01 X_Z_F_

G01 X_F_

G01 Z_F_

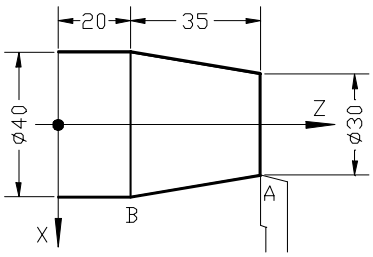
Description:

(1) When the processing starts each time, it's in the G01 status automatically.

(2) The non-movement coordinate requires no programming..

(3) The coordinate of target point can be written with absolute value or increment value..

(4) When G01 is processed, its feed speed implements according to given F value, and F scope is: 1~6000 mm/min..



Illus 2-2 G01

(5) G01 can also be written as G1..

Example: The program of right figure is as follows: (supposing tool nose is at A point).

Absolute value mode programming: G01 X40 Z20 F150.

Incremental mode programming: G91 G01 X10 Z-35 F150

2.2.3 G02—Interpolation of clockwise arc

Format: G02 X_Z_I_K_F_

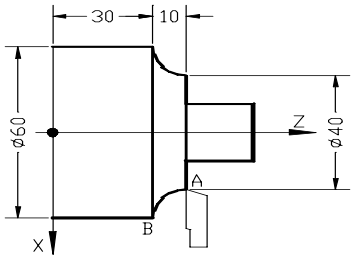
G02 X_Z_R_F_

Description:

(1) When X and Z are at G90, the end coordinate of arc is the absolute coordinates value which is relative to programming zero.. When it's at G91, the arc end is the increment value of relative arc start point.. Towards to G90, G91, I and K, they are all the coordinate values of center relative arc start point, and I is the diameter amount in X direction value, K is Z direction. The circle center coordinate can't be omitted at the circular interpolation, unless it's programmed with R (arc radius).

(2) When G02 instruction is programmed, it can directly program over the quadrant circle and full circle and so on (R programming can't be used for full circle).

Note: When passing quadrant, . it can automatically implement clearance compensation, if the clearance compensation isn't be input in the parameter region, or the clearance compensation in the parameter region differs much to the practical reverse clearance of machine tool, which will generate obvious cut mark on the work piece.



Illus 2-3 G03

(3) The full circle can't be programmed with R..

(4) R is the arc's radius which is the number with symbol, "+" signifies that the arc angle is less than or equal to 180 degree; "-" signifies that the arc angle is greater than 180 degree..

(5) G02 can also be written as G2.

Example: The AB segment arc program of processing right figure is as follows:

Absolute value mode:

G90 G02 X60 Z30 I20 K0 Fl50(circle center coordinate programming)

G90 G02 X60 Z30 R10 F150(radius R programming)

Incremental mode:.

G91 G02 X20 Z-10 I20 K0 Fl50(circle center coordinate programming)

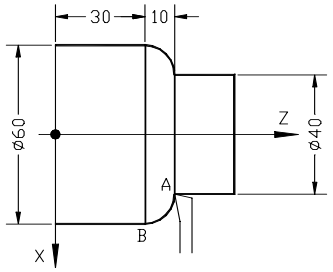
G91 G02 X20 Z-10 R10 F150(radius R programming)

2.2.4 G03—Inverse arc interpolation

Format: **G03 X_Z_I_K_F_**

G03 X_Z_R_F_

Description: When programming with G03 instruction, except that the arc rotating direction is opposite, the rest are same as the G02 instruction..



Illus 2-4 G04

Example: The program of right figure is as follows:.

Absolute value mode:.

G90 G03 X60 Z30 I0 K-10 F100

(circle center coordinate programming)

G90 G03 X60 Z30 R10 Fl00(radius R programming)

Incremental mode:.

G91 G03 X20 Z－10 I0 KF100(circle center coordinate programming)

G91 G03 X20 Z－10 R10 F100 (radius R programming)

2.2.5 G04—Time delay

Format: **G04 K×××. ××**

Description: After the program delays post-K programming value (second), continue to operate, and the scope of time delay is from 0.01 second to 65.5 seconds.

2.2.6 G09—Feed angle-specified stop

Format: **G09**

Description: G09 is used to check whether servo is in place, and because servo has following errors, when it reaches setting error limit, it has some minute time (related with feed speed). When servo following error is less than given value, servo will send one angle-specified stop signal (XPSN, YPSN and ZPSN) to CNC system. After the system completes certain segment program, if this segment has G09, then CNC will check whether various axes have PSN signal input within a period of time, and this time is set by 89# parameter. If there is no signal when set time is exceeded, CNC will give 57# alarm to continue operation.

2.2.7 G20—Sub-program call

Format: **G20 N ××. ×××**

Description:

(1) The first 2-digit after N is the program name of sub-program which will be called and 2-digit is allowed. The 3-digit after decimal signifies the cycle number of this call which can be from 1 to 255..

(2) The parameter in the sub-program must be assigned definite numerical value by P when it's called by G20..

(3) This segment of program mustn't appear contents outside of the descriptions above.

(4) Various sub-programs can repeat embedding call for 10 times, but calling itself is prohibited.

2.2.8 G22—Sub-program definition

Format: **G22 N ××**

Escription: (1) The sub-program name is initiated with N, and two digits after N is the sub-program name.

(2) G22 NXX mustn't be in the same segment with other instructions.

(3) G22 and G24 appear in pair, which forms one complete sub-program body.

(4) The parameter data in the sub-program has two kinds of formats:

A) Constant format, the data is constant set by programming, namely 0~9.

B) Parameter format, the numeric section such as function number, parameter and so on in the program can be signified with variable, while the specific value of variable is imported by P=×× definition in the main porgram of calling sub-program. This system can process 10 variable parameters: P0 P1 ···P9.

(5) Sub-program and transition processing (G25 and G26) can implement compound nestification for most 10 times.

(6) When it's necessary for parameter to define variables, P0 = ××, P1 = ×× and so on can be used to assign definite numerical values to 0#-9#. No matter whether P appears in the main porgram or sub-program, this parameter will be superseded with the lately assignment.

2.2.9 G24—Completion and returning of sub-program

Format: **G24**

Description:

(1) G24 signifies completion of program and returns to the next segment of program calling this sub-program..

(2) G24 and G22 appear in pair..

(3) The segment of G24 disallows other instructions to appear..

Example: The parameter transmission process in the subprogram call will be described through the following example, please apply it..

Main program P01.

```
N0010  S1000  M03
N0020  P7=200  P8=50  P9=02
N0030  G20  N05
N0040  M02
```

Sub-program N05

```
N0010  G22  N05
N0020  G92  X50  Z100
N0030  G01  X40  FP7
N0040  Z97
N0050  GP9  Z92  X50  I10  K0  FP8
N0060  G01  Z-25  FP7
N0070  G00  X60
N0090  Z100
N0100  G24
```

Note: (1) If P parameter is not defined when subprogram is called, then the value of P parameter in the subprogram is indefinite.

（2） The parameter can also be used in the main program.

2.2.10 G25—Jump processing.

Format: G25 N××××. ××××. ×××

Description: (1) The cycle body which is defined by this format is the defined block (including these two segments) between two block numbers following N, and the digit defines the call number of this block, from 1 to 255, and 1 will be considered without compiling..

(2) The next segment processing program after G25 instruction completes is the one of jump processing block..

(3) Other instructions are prohibited in the G25 block..

Example: **N0010 G92 X50 Z100**
N0020 G25 N0040.0060.02
N0030 G00 X10 Z20
N0040 G01 X40 Z80 F300
N0050 Z60
N0060 G00 X50 Z100
N0070 G04 K3
N0080 M02

The processing sequence of program above is:

N0010-N0020-N0040-N0050-N0060-N0040-N0050-N00-N0070-N0080

2.2.11 G26—Transition processing (sub-program call in the program)

Format: **G26 N××××. ××××. ×××**

Description: The transition processing instruction completes, the next processing section is the next one of G26 N××××. .××××.. ×××, which is the difference from G25, and the rest are same as G25.

Example: **N0005 S800 M03**
N0010 G26 N0050.0080.02
N0020 G04 K2
N0030 G01 X2 F20
N0040 G00 X0 Z0
N0050 G92 G90 X0 Z0
N0060 G01 Z-20 X20 F300
N0070 M00
N0080 Z-40
N0090 Z-60 X0
N0100 M02

The processing sequence of program above is:

N0005-N0010-N0050-N0060-N0070-N0080-N0050-N0060-N0070-N0080
N0020-N0030-N0040-N0050-N0060-N0070-N0080-N0090-N0100

2.2.12 G27—Endless cycle

Format: **G27 N××××. ××××**

Description:

(1) The block between the first and second block numbers is the interval of endless cycle, once entering G27 status, the system will infinitely repeat carrying out the operating track which is defined by this block.

(2) In order to guaranty that the coordinate doesn't offset when every cycle starts, the block is required to be the sealed track, else the start point will shift at every start, and finally exceeds working table.

2.2.13 G30—Magnification and minification cancel

Format: **G30**

Description: When the G31 magnification and minification is implemented, G30 cancels effect of G31.

2.2.14 G31—Magnification or minification

Format: **G31 K××.×**

Description: (1) The scope of multiply factor is 0.001-65.5, namely K0.001 ~ K65.5..

(2) The effect of multiply factor is to magnify or minify K times for the various segments' size of processing track evenly.

(3) The multiply factor has no effect on the cutting tool.

2.2.15 G47—Short line speed auto-transition

Format: **G47**

Description: When processing non-circle curved plane, the general software of CAD/CAM uses very short line to impend curved plane under precondition of guarantying certain precision, and the numerical control system controls various coordinate axes to regulate speed between two segments of lines. Transiting from one segment of line to the next one under precondition of guarantying cutting linear speed unchanged, which avoids uneven linear speed owing to raising speed and reducing speed of each short line, accordingly reducing shake of machine tool and enhancing actual cutting speed and surface smoothness. When G47 is valid, only the two segments of lines must conform the following conditions can they transit automatically:

(1) Line length is less than 21 mm

(2) The intersection angle of two lines is less than 20 degree

2.2.16 G48—Cancel G47.

Format: **G48**

Description:G47 and G48 are the option function, the numerical control system doesn't include this function unless it's especially specified.

2.2.17 G53—Cancel zero bias

Format: **G53**

Description:

(1) After zero bias, G53 function will restore the processing element programming zero to the initial one.

(2) G53 function will be effective after implementing zero bias function.

2.2.18 G54—Absolute zero bias

Format: **G54 X_Z_**

Description:

(1) G54 function parallel moves programming origin to the specified X'O'Z' coordinate position..

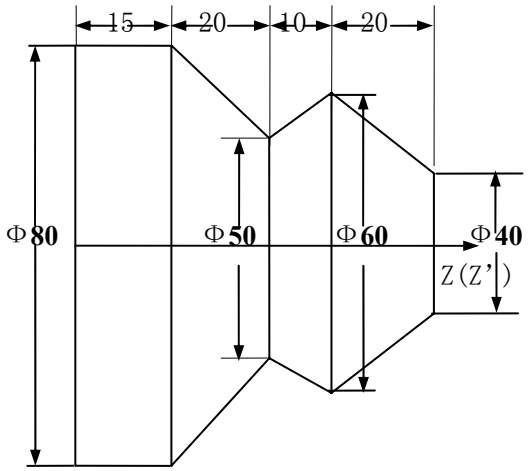
(2) The three coordinates of X and Z can all parallel move, it can also for one of coordinates to parallel move, and the coordinates which aren't compiled don't parallel move..

(3) G54 function is the independent block, and other instructions are prohibited..

(4) The post G54 block will be compiled on the base of G54 established new coordinate system without considering the original coordinate system effect..

(5) When processing, the dynamical coordinate display is still relative to original coordinate system origin..

(6) G54 itself is not the movement instruction, it only memorizes coordinate bias. If it's necessary for cutting tool to operate to G54, the G01 or G00X0Z0 block must be compiled once more to make cutting tool operate to G54.



Example:

```
N0010  G92  G90  X40  Z65
N0020  G01  X60  Z45  F100  (AB)
N0030  G54  Z35
N0040  G01  X50  Z0  (BC)
N0050  X80  Z-20  (CD)
N0060  G53
N0070  M02
```

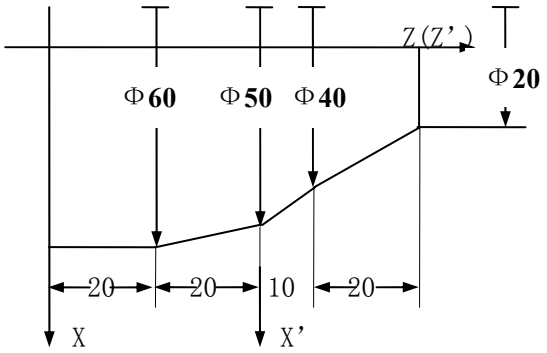
2.2.19 G55—Increment zero bias

Format: **G55 X_Z_**

Description:

(1) G55 function will make coordinate system origin parallel move to XZ in increment from the cutting tool current position, which forms new coordinate system..

(2) Other cautions are same as G54.



Example:

```
N0010  G90  G92  X20  Z70
N0020  G01  X40  Z50  F100
N0030  G55  Z-10
N0040  G01  X50  Z0
N0050  X60  Z-20
N0060  G53
N0070  M02
```

2.2.20 G56—Current point bias

Format: **G56**

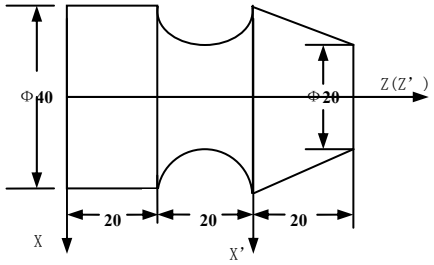
Description:

(1) G56 function will set the cutting tool current position as the coordinate origin, the posterior programming will regard this point as coordinate origin without considering the original coordinate system effect.

(2) The rest is same as G54.

Example:

```
N0010  G90  G92  X20  Z60
N0020  G01  X40  Z40  F100
N0030  G56
N0040  G02  Z-20  10  K-10
N0050  G53
N0060  M02
```



2.2.21 G71—Internal (external) diameter cutting compound cycle

Format: **G71 I_K_N_X_Z_F_**

Description: The fine machining path of rough machining and fine machining showed in the instruction execution figure 1 is the track of A→B→C→D..

Among them: I: cutting depth (cutting amount for each time), the symbol isn't added when specified, and the direction is determined by vector AB:.

K: Retract amount for each time, the symbol isn't added when specified, and the directions of X and Z are respectively determined by X (X-direction fine machining allowance) and Z (Z direction rough machining allowance);

N: Fine machining block number;

X: X direction fine machining allowance;.

Z: Z direction fine machining allowance;

F: The F in the G71 programming is valid at the time of rough machining, and the F in the fine machining block is valid at the time of fine machining.

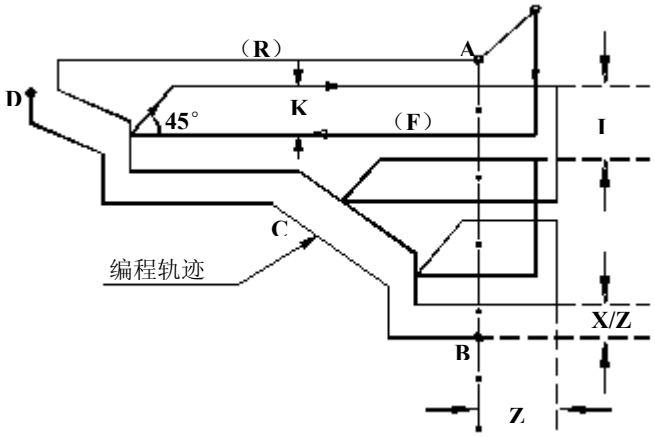


Figure 1 Internal (external) diameter cutting compound cycle G71.

Under G71 cutting cycle, the cutting feed direction parallels to Z axis, and the symbols of X and Z are showed as Figure 2..（+） signifies movement along axis positive direction, and (-) signifies movement along axis negative direction..

Note: (1) (fine machining block number) must be greater than 1;

(2) A→B must be completed by G00 instruction, and the G00 instruction can't be included within B→C→D;.

(3) The Z direction movement amount shouldn't be in the A→B block, X direction movement amount is equal to X direction movement total of B→C→D.

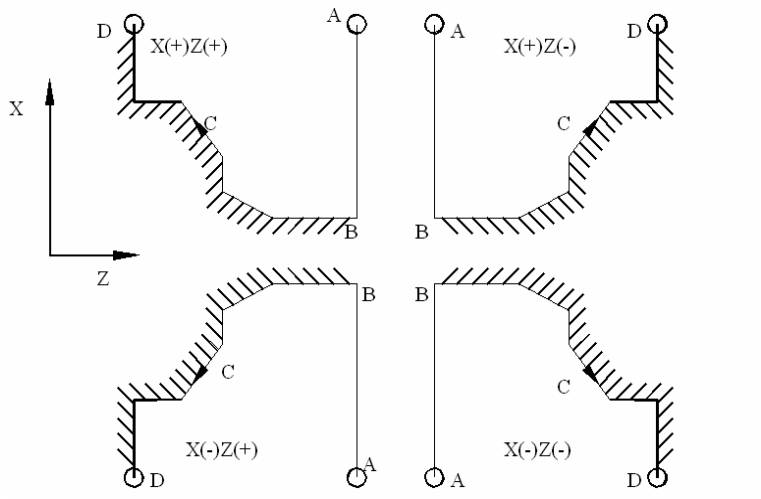


Figure 2 Symbols of X and Z under G71 compound cycle

2.2.22 G72—End surface cutting compound cycle

Format: **G72 I_K_N_X_Z_F_**

Description: The fine machining path of rough machining and fine machining showed in the instruction execution figure 3 is the track of A→B→C→D..

Among them: I: cutting depth (cutting amount for each time), the symbol isn't added when specified, and the direction is determined by vector AB:.

K: Retract amount for each time, the symbol isn't added when specified, and the directions of X and Z are respectively determined by X (X-direction fine machining allowance) and Z (Z direction rough machining allowance);.

N: Fine machining block number;.

X: X direction fine machining allowance;.

Z: Z direction fine machining allowance;.

F: The F in the G72 programming is valid at the time of rough machining, and the F in the fine machining block is valid at the time of fine machining..

Under G72 cutting cycle, the cutting feed direction parallels to X axis, and the symbols of X and Z are showed as Figure 4.. （+） signifies movement along axis positive direction, and (-) signifies movement along axis negative direction..

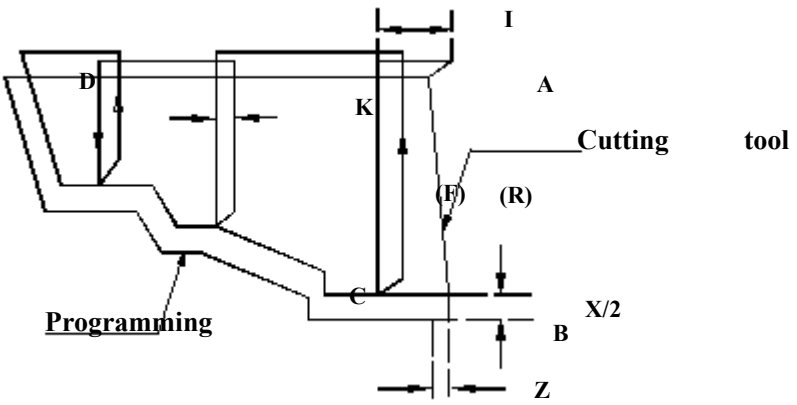


Figure 3 End surface cutting compound cycle G72.

Note: (1) N (fine machining block number) must be greater than 1;.

(2) A→B must be completed by G00 instruction, and the G00 instruction can't be included within B→C→D;.

(3) The X direction movement amount shouldn't be in the A→B block, Z direction movement amount is equal to Z direction movement total of B→C→D..

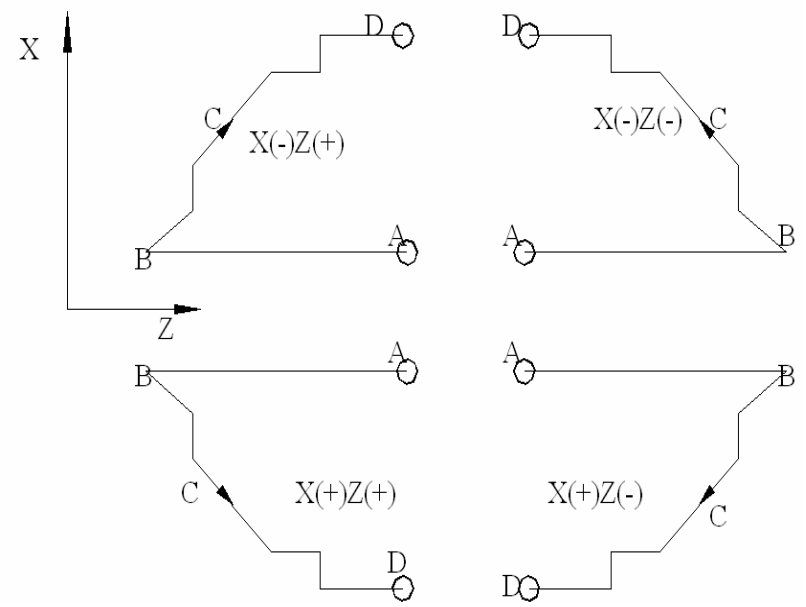


Figure 4 The symbols of X and Z under G72 end surface cutting compound cycle

2.2.23 G73—Sealed contour compound cycle

Format: G73 I_K_N_L_X_Z_F

Description: The cutting tool track showed in Figure 5 is the closed loop when this function is of cutting work, and the cutting tool feeds gradually, which makes the closed cutting loop gradually approach to the element final shape and finally cut into the shape of work piece, and its fine machining path is the track of A→B→C→D..

This instruction can implement highly-efficient cutting to the preliminary-finished work piece in the rough machining such as casting, forging and so on.

Among others:

I: The rough machining total allowance in the X direction;

K: The rough machining total allowance in the Z direction;

N: Fine machining block number;

L: Rough cutting number;

X: X direction fine machining allowance;

Z: Z direction fine machining allowance;.

F: The F in the G73 programming is valid at the time of rough machining, and the F in the fine machining block is valid at the time of fine machining..

Note: I and K signify the total cutting amount of rough machining, and the rough cutting number is L, then the cutting amounts in the X and Z directions for each time are I / L and K/L;.

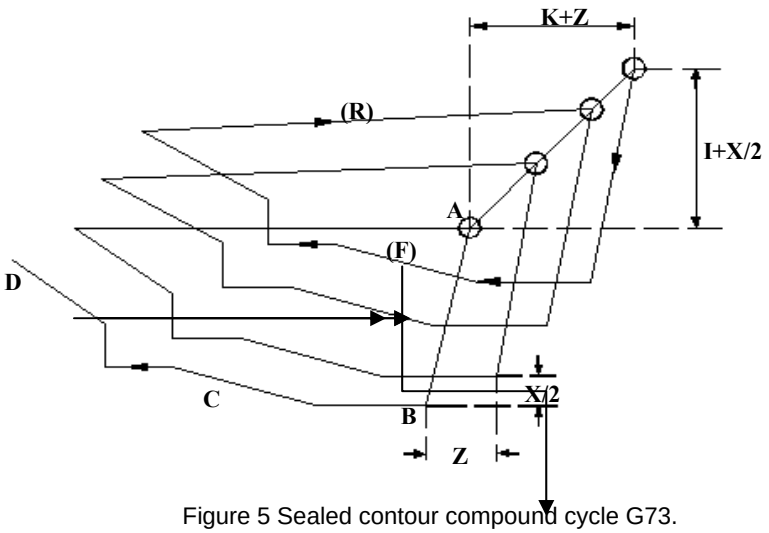


Figure 5 Sealed contour compound cycle G73.

Appendix: Examples of various compound cycles..

% P55

N0010 G00 X260 Z370
N0020 G71 I7 K2 N8 X.37 Z.29 F2400
N0030 G90 X50 Z370
N0040 G 01 X50 Z350 F240
N0050 G 01 X100 Z230
N0060 G 01 X100 Z170
N0070 G 01 X160 Z170
N0080 G 01 X160 Z100
N0090 G 03 X210 Z50 I0 K-50
N0100 G 02 X260 I50 K0
N0110 G 00 X260 Z370
N0120 G27 N0020.0110
N0130 M02

% P66 Diameter programming

N0010 G 00 X520 Z370 F2400
N0020 G71 I14 K2 N8 X.74 Z.29 F2400
N0030 G00 X100 Z350 F240
N0040 G01 X100 Z350 F240
N0050 G01 X220 Z230
N0060 G01 X200 Z170
N0070 G01 X320 Z170
N0080 G01 X320 Z100
N0090 G03 X420 Z50 I0 K-50
N0100 G02 X520 Z0 I100 K0
N0110 G00 X520 Z370
N0120 G27 N0020.0110
N0130 M02

% P59 Cutting internal diameter

```
N0010  G00  X10  Z370
N0020  G71  17  K2  N8  X-.37  Z.29  F2400
N0030  G00  X240  Z370
N0040  G02  X190  Z320  I0  K-50  F240
N0050  G03  X140  Z270  I-50  K0
N0060  G01  X100  Z200
N0070  G01  X100  Z150
N0080  G01  X60  Z150
N0090  G01  X60  Z80
N0100  G01  X60  Z80
N0110  G00  X10  Z370
N0120  G27  N0020.0110
N0130  M02
%P77
N0010  G00  X260  Z370
N0020  G72  I7  .K2  N8  X.37  Z.29  F2400
N0030  G00  X260  Z40
N0040  G01  X160  Z140
N0060  G01  X160  Z210
N0070  G01  X100  Z210
N0080  G01  X100  Z270
N0090  G03  X50  Z320  I0  K50
N0095  G02  X0  Z370  I-50  K0
N0100  G00  X260  Z370
N0110  G27  N0020.0100
N0120  M02
%P88  Diameter programming
N0010  G00  X 520  Z370
N0020  G72  I7  K2  N8  X.37  Z.29  F2400
N0030  G00  X520  Z40
N0040  G01  X500  Z90  F240
N0050  G01  X320  Z140
N0060  G01  X320  Z210
N0070  G01  X200  Z210
N0080  G01  X200  Z270
N0090  G03  X100  Z320  I0  K50
N0095  G02  X0  Z370  I-100  K0
N0100  G00  X520  Z370
N0110  G27  N0020.0100
N0120  M02
%P99
N0010  G00  X270  Z390
N0020  G73  I220  K60  N6  L20  X.6  Z.3  F2400
N0030  G00  X40  Z370
```

```
N0040  G01  X40  Z270  F240
N0050  G01  X75  Z140
N0060  G02  X175  Z40  I100  K0
N0070  G00  X200  Z20
N0080  G01  X260  Z390
N0100  G27  N0020.0090
N0110  M02
％P98  Diameter programming
N0010  G00  X540  Z390
N0020  G73  1440  K60  N6  L20  X.6  Z.3  F2400
N0030  G00  X80  Z370
N0040  G01  X80  Z270  F240
N0050  G00  X150  Z140
N0060  G02  X350  Z40  I200  K0
N0070  G01  X40  Z20
N0080  G01  X520  Z00
N0090  G00  X540  Z390
N0100  G27  N0020.0090
N0110  M02
```

It's necessary to specify the fine machining path and rough machining engagement when this group of compound cycle instructions are operated, then the system will automatically calculate rough machining path and cycle number.

2.2.24 G74—Returning to reference point (mechanical origin)

Format: **G74 X_Z_**

Description:

(1) Other contents in this segment are prohibited..

(2) The coordinates following G74 will return to zero in turn with X, Z sequence..

(3) The reference point switch must be confirmed to be installed on the machine tool before using G74.

2.2.25 G75—Returning to presetting cutter point from reference point

Format: **G75 X_Z_**

Description:

(1) Other contents in this segment are prohibited.

(2) After G75 instruction completes, the small coordinates XP and ZP return to position set by 8# and 9# parameters.

(3) After G75 completes, the big coordinate will be automatically set as 18# and 19# specified values.

2.2.26 G76—Returning to processing start point from current position (feed point)

Format: **G76 X_Z_**

Description:

(1) Other contents in this segment are prohibited..

(2) The coordinate which is relative to origin on the machine tool is displayed with big

coordinate, and the coordinate of certain position can be memorized in 18# and 19# parameters. This function can return to this position from machine tool optional position, and the speed is same as G00..

(3) The processing start point (18# and 19#) is set referring to processing origin (like chuck center), and the result of G76 execution is to move tool nose to the same coordinate position with parameters 18# and 19#.

2.2.27 G77—Recovering current coordinate system

Format G77 X_Z_

G77 X_

G77 Z_

Description: G77 can only be used with G75 to recover work piece coordinate system before G75 execution.

2.2.28 G81—Excircle (inner circle) fixed cycle

Format: G81 X_Z_R_I_K_F_

Description:

(1) Under absolute coordinates mode, X and Z are absolute coordinates of another end surface (end point), and under incremental programming mode, X and Z are increment values of end point which is relative to current position..

(2) R is the processing diameter of start point section..

(3) I is the rough turning feed amount and K is the fine turning feed amount. I and K are symbolic numbers, and their symbols should be the same. . The symbols are specified as follows: Cutting from outward central axis (turning excircle) is "-", on the contrary it's "+"..

(4) Various X, Z and R values determine different shapes of excircle such as: with taper or without taper, positive taper and negative taper, left cutting or right cutting and so on..

(5) F is the feed speed (mm/min) of cutting processing..

(6) After processing completes, the cutting tool stops at the end point.

Example 1: positive taper excircle, the left cutting is implemented.

G90 G81 X40 Z100 R30 I-1 K-0.2 F200(Absolute value programming)

G91 G81 X0 Z-50 R30 I-1 K-0.2 F200(Increment programming)

Processing course:.

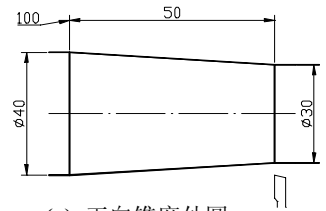
① G01 feeds two times of I (the first cutting is I, and the final cutting is I+K fine turning), implementing depth cutting;.

② G01 two axes interpolation, cutting to the end point section, and it will stop if processing completes;.

③ G01 retracts I to safety position, and the auxiliary section smoothness processing is implemented simultaneously..

④ G00 retracts △Z to start point section;.

⑤ G00 fast feeds to I position off from work piece surface, and I is preserved to implement the next step cutting processing and repeated to ①..

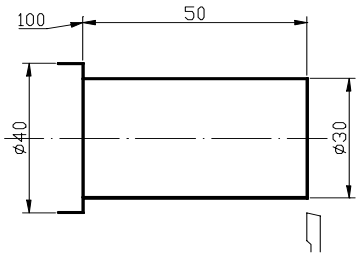


(a) 正向锥度外圆

(b) 加工过程图

图 **2-14**

Example 2: Non-taper excircle, the left cutting is implemented.



G90 G81 X30 Z100 R30 I-1 K-0.2 F200 (Absolute value programming)

G91 G81 X-10 Z-50 R30 I-1 K-0.2 F200 (Increment programming)

Example 1 of processing course.

Example 3: Negative taper excircle, the left cutting is implemented.

Taking into consideration of cutting amount of end point, the cutting tool should maintain proper distance (≥△Φ) from work piece at the start point..

G90 G81 X30 Z100 R40 I-1 K-0.2 F200 (Absolute value programming)

G91 G81 X-30 Z-50 R40 I-1 K-0.2 F200 (Increment programming)

Processing course:

① G01 feeds two times of I (the first cutting is I, and the final cutting is I+K fine turning), implementing depth cutting;.

② G01 two axes interpolation, cutting to the end point section, and it will stop if processing completes;.

③ G01 retracts I, and the auxiliary section smoothness process is implemented simultaneously;.

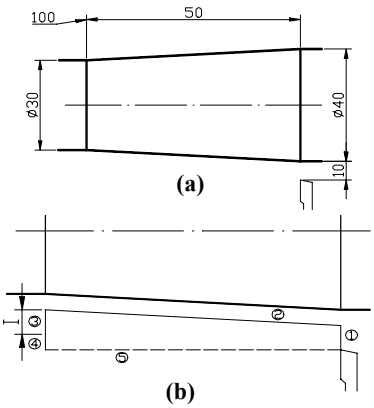


图 **2-16**

④ G00 fast retracts △Φ to safety position;.

⑤ G00 fast retracts to start point section and is repeated to ①.

Example 4: Negative taper excircle, the right cutting is implemented.

Absolute value programming:

G90 G81 X40 Z150 R30 I-1 K-0.2 F200

Increment programming:

G91 G81 X0 Z50 R30 I-1 K-0.2 F200

The processing course is same as example 1.

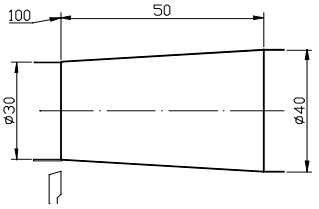


图 **2-17**

Example 5: Negative taper inner circle, the left cutting is implemented.

G90 G8l X30 Z100 R40 I l K0.2 F200 (Absolute value programming)

G91 G81 X30 Z-50 R40 I1 K0.2 F200 (Increment programming)

Processing course:.

It's similar to example 1, and the difference is that the cutting tool retracts to the central axis direction when retracting.

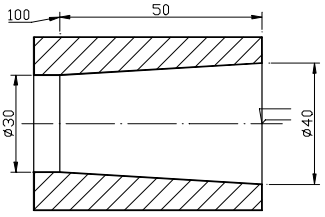


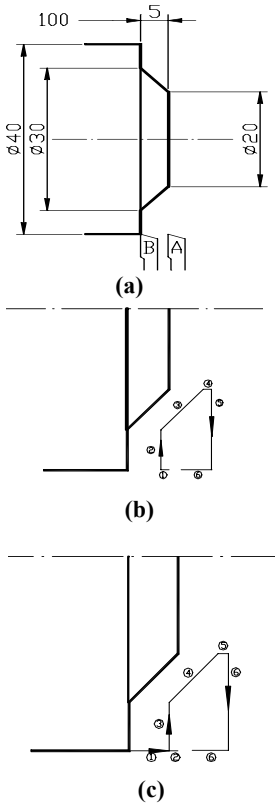
图 **2-18**

2.2.29 G82－End surface fixed cycle

Format: G82 X_Z_R_I_K_F

Description:

(1) Under absolute coordinates mode, X and Z are absolute coordinates of another end surface (end point), and under incremental programming mode, X and Z are increment values of end point which is relative to current position..



Illus 2-19

(2) R=（end point diameter—start point diameter),the end point (start point) diameter is the final cone diameter on the end point (start point) section. When the flat end surface cutting short work piece processes, the end-point diameter and start point diameter are both zero. . R is the symbolic number, "+" signifies that end-point diameter is greater than start point diameter, and "-" is the opposite..

(3) I is the rough turning feed and K is the fine turning feed. I and K are symbolic numbers, and their symbols should be the same. The symbols are specified as follows: leftwards cutting processing is "+" (omission is allowable), and rightwards cutting processing is "-"..

(4) Various K, Z, R and I values determine different shapes of end surface, among them R value determines whether there is taper, and end surface has no taper when R is equal to 0; If X=0 and R=0 at the same time under absolute mode, then the work piece will be cut short and end surface is turned flat; the symbol of R determines taper direction with taper end surface; all symbols of Z, R and I determine the cutting mode of taper end surface, which is divided into internal cutting and external cutting. . Towards to programming of various conditions, they will be elaborated on with examples..

(5) F is the feed speed (mm/min) of cutting processing..

(6) Towards to the end surface without taper, the processing length has no limit; but towards the end surface with taper, the processing length is limited within the one between two end surfaces, and cutting tool requires stopping on one of two end surfaces before processing starts. After processing completes, the cutting tool stops at the programming end point..

Example 1: The end surface cycle which taper has steps and the cutting tool can stop at A or B position and starts from A position at the beginning..

G90 G82 X30 Z100 R10 I—1 K—0.2 F200 (Absolute value mode)

G91 G82 X—10 Z—5 R10 I—1 K—0.2 F200 (Increment mode)

Processing course:.

① G01 and Z direction feed two times of I (the first cutting is I, and the final cutting is I+K fine turning), G1 and X direction feed to the end point implementing depth cutting;

③ G01 two axes interpolation, cutting to the other end surface; G01 and Z direction retract I to safety position, and the auxiliary section smoothness process is implemented simultaneously;

⑤ G00 and X direction retract to start point;.

⑥ G00 and Z direction fast feed to I position off from work piece surface, and I is preserved to implement the next step cutting processing..

⑦ If the processing completes, G01 will feed to end point and stop, else it'll repeated to ①..

Starting from B position.

G90 G82 X20 Z105 R-10 I-1 K-0.2 F200 (Absolute value mode)

G91 G82 X-20 Z5 R-10 I-1 K-0.2 F200 (Increment mode)

Processing course:.

① G00 and Z direction fast feed to A position;.

② G01 and Z direction feed two times of I (the first cutting is I, and the final cutting is I+K fine turning), implementing length cutting;.

③ G01 and X direction feed to the end point implementing depth cutting;.

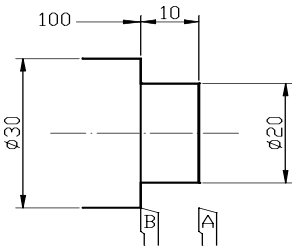
④ G01 two axes interpolation, cutting to the other end surface;.

⑤ G01 and Z direction retract I to safety position, and the auxiliary section smoothness processing is implemented simultaneously..

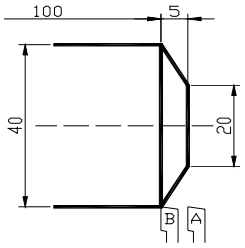
⑥ If the processing completes, G01 will feed to end point and stop, else G00 and X direction retract to start point;.

⑦ G00 and Z direction fast feed to I position off from work piece surface, and I is preserved to implement the next step cutting processing..

⑧ Repeated to ②..



Illus 2-19



Illus 2-20

Example2: The end surface cycle which taper has no steps and the cutting tool can stop at A or B position and starts from A position at the beginning.

G90 G82 X20 Z100 R0 I—1 K—0.2 F200 (Absolute value mode)

G91 G82 X—10 Z—10 R0 I—1 K—0.2 F200 (Increment mode）

Processing course: It's similar to example 1.A, and the difference is without the ③ step..

Starting from B position.

G90 G82 X20 Zll0 R0 I—1 K—0.2 F200 (Absolute value mode)

G91 G82 X—10 Z10 R0 I—1 K—0.2 F200 (Increment mode)

Processing course: It's similar to example 1.B, and the difference is without the ④ step..

Example 3: The end surface cycle which taper has no steps, the cutting tool can stop at A or B position and starts from A position at the beginning..

G90 G82 X40 Z100 R20 I -1 K -0.2 F200 (Absolute value mode)

G91 G82 X0 Z -5 R20 I -1 K -0.2 F200 (Increment mode)

Processing course: It's similar to example 1.A, and the difference is without the ② step..

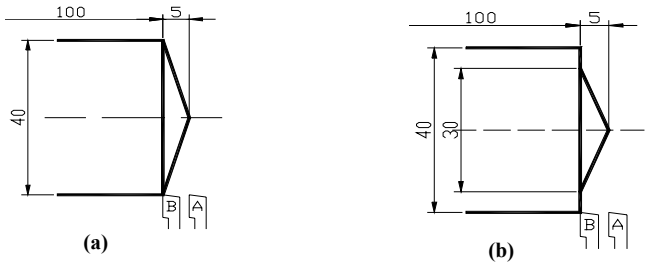
Starting from B position.

G90 G82 X20 Z105 R -20 I -1 K-0.2 F200 (Absolute value mode)

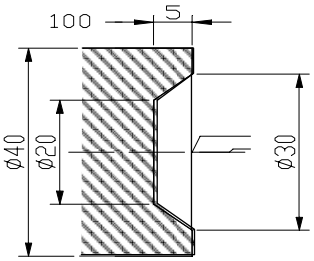
G91 G82 X -20 Z5 R -20 I -1 K-0.2 F200 (Increment mode)

Processing course: It's similar to example 1.B, and the difference is without the ③ step..

According to various X and R values, the following figure can also be programmed:



Illus 2-21



Example 4: The concave end surface cycle with steps. The cutting tool should stop at the central position when it's processed from center to outside, which guaranties the cutting tool not to be bumped at the time of retracting..

G90 G82 X20 Z100 R—10 I—1 K—0.2 F200 (Absolute value mode)

G91 G82 X20 Z—5 R—10 I—1 K—0.2 F200 (Increment mode)

In the examples above, if I and K are changed into "+" simultaneously, then the figure will regard X-axis as mirror image and right processing will be implemented.

2.2.30 G85—Threading cycle

31DT standard configuration has no this function

2.2.31 G86—Metric thread cycle

Format: G86 ΔX Z K I/D R N L J

格式：G86 X Z_ K_ I/D_ R_ N_ L_ J_

Z direction de-trailing length (Plus positive value).

Cycle number

Thread head number, which is used for multiple thread (N≤150)

The diameter difference between thread outside diameter and root diameter, which is positive value

The de-trailing length after thread is completed in X direction/screw in distance value

Thread pitch (metric system Kmm, English system K teeth/Inch).

Length of thread, it can be absolute or relative programming

ΔX is the diameter change in X direction, and it's the straight thread when ΔX is equal to 0.

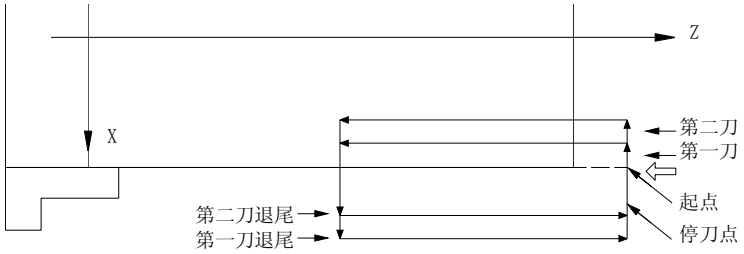
Description:

(1) The feed depth of each time is determined by P10 and P11assignment statements before program, and the X direction single side feed smooth thread flat of final cutting.. （smoothing tool amount is determined by 25# parameter).

(2) The de-trailing direction of thread in X direction is determined by I value, "+" is the external thread and "-" is the internal thread..

(3) The initial position of thread processing cycle is where the tool nose is aligned with thread top diameter..

(4) J value signifies the preserved de-trailing length in Z direction. . Non-compiling signifies the system default..



Illus 2-24

(5) When it's necessary to wait for thread pitch ending, the screw out function can be used, and it's format is to add J value into the general G86 function. The digits after J signifies de-trailing length in Z direction, namely when the distance between tool nose and thread end-point is the value after J (Z direction), the de-trailing will be started in X direction. When J value isn't compiled, the de-trailing will be implemented after Z direction moves to approach end-point..

(6) Normally the de-trailing length is signified with value after I in X direction of G86, when D is compiled, it signifies that the thread feed is screw in. Please note when using screw in function: When feed is started, the tool nose must be in the distance greater than or equal to D value outside of work piece surface, else the cutting tool will be bumped; the screw in distance is equal to de-trailing one (X direction)..

(7) According to I, J and D programming, there are four combinations:.

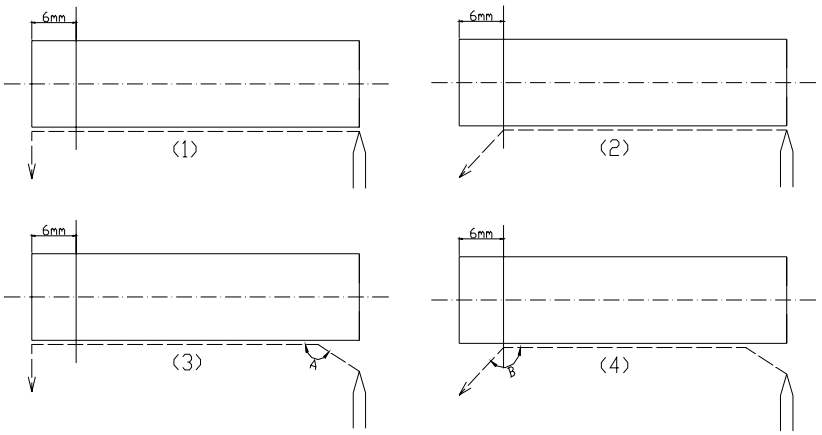
a) G86 Z—100 R2 K3 L10 I5 common thread processing

b) G86 Z—100 R2 K3 L10 I5 J6 de-trail with 6mm from Z direction end-point

c) G86 Z—100 R2 K3 L10 D5 screw in cut-in but without equal thread pitch de-trailing

d) G86 Z—100 R2 K3 L10 D5 J6 screw in and screw out

The angles of A and B are different according to various thread processing speed (K× n rotating speed × thread pitch), screw in and screw out speed in X direction (16# parameter, and 16# parameter ⩽59# parameter), screw in and screw out time constant in X direction (49# parameter) and Z direction time constant (400 parameter) when the thread is processed..



Illus 2-25

(8) The 16# parameter signifies the screw in and screw out speed in X direction, and the general programming value is from 2000 mm／min to 3000 mm /min. However when the set value is less than 1000 mm/min, it'll be automatically set to 2500 mm/min when system processes..

(9) Towards to the sub-cutting tool of thread cutting, it can be optionally set in the program, adopting the assignment statement P10=0, 1 and 2. . a). When P10=0, it's the equidistance feed at the time of thread cutting, namely the feed amount of each time is R／L, and when the 25# parameter is not equal to 0, finally increasing one cut of smoothing tool. .b). When P10=1, it's the equal cutting amount feed. . Towards to 60 degree screwer, the metal cutting amount of every feed can be guarantied basically identical. . Its formula is: △Rn=($\sqrt{n}$ -$\sqrt{n-1}$) × R/$\sqrt{L}$, △Rn: Feed amount of the n time. . N: Feed of the n time. . N≤L.. L: Cycle number, R: Total cutting depth. c) When P10=2, it's the equal cutting amount feed, and the first cut when P10=1 is divided into two cuts for cutting. . If the cutting amount of first cut is too big, P10 can be set as 2, and system will divide the first cut into two for cutting, to prevent damaging tool nose such as when R=1.0, L=5:.

P10	The first cut	The second cut	The third cut	The fourth cut	The fifth cut	The sixth cut	The seventh cut
0	0.2	0.2	0.2	0.2	0.19	0.01..	/
1	0.45	0.19	0.14	0.12	0.09	0.01	/
2	0.23	0.22	0.19	0.14	0.12	0.09	0.01

(10) Towards to the thread cutting mode, it can be optionally set in the program, adopting the assignment statement P11=0, 1, 2 and 3.. A） When P11=0, it's the normal cutting, b）When P11=1, it's the left cutting, c) When P11=2, it's the right cutting, d) When P11=3, it's the left and right cutting..

Example: N0010 P10=2 P11=0
N0020 G00 X100 Z100
N0030 G00 X50 Z1
N0040 G86 Z50 K1 R1 I6 L5
N0050 G00 X100 Z100
N0060 M02

(11) There is the course of speed increase and decrease when the thread processing is started and completed, and the thread is inaccurate within this period, accordingly these two regions must be avoided at the time of actual processing. . The 40# parameter defines the speedup time constant in Z direction when the thread is processed..

(12) The speed of stepping motor/servomotor shouldn't exceed certain value when the thread is processed, such as 2.5m/min, and this speed is related with machine tool size and power of motor, which is determined by 45# parameter..

(13) CNC measures the spindle rotating speed before thread is cut, and determines the optimal course of speed increase and decrease of stepping motor, as well as judges whether spindle rotating speed is stable, after the zero signal of encoder appears, the processing will be started, and this course requires 50—100 millisecond. If the spindle rotating speed is not stable, CNC will start processing after the spindle rotating speed is stable. . If the steady speed isn't measured, the thread processing won't be implemented in general. 23# parameter signifies percentage of spindle speed fluctuation, which 10-13 is used in normal, and the spindle rotating speed fluctuation ratio should be less than or equal to 23# parameter at actual processing.. (14)

The 25# parameter sets the smoothing tool amount of the final cut of thread, and if thread processing cycle requires no smoothing tool finally, the 25# parameter should be equal to 0..

(15) When only G86X-Z-K is compiled, the single-tool thread cutting will be implemented without retracting after cutting is completed..

Note: At screw in (D value is valid), only straight thread can be processed. . J must be positive value.

2.2.32 G87—English system thread cycle.

Format: Same as G86..

Note: The thread pitch is K teeth/Inch.

2.2.33 G90—Programming with absolute value mode

Format: G90

Description:

(1) When G90 is compiled into program, the programming zero will be regarded as benchmark for all the compiled coordinate values afterwards..

(2) After the system is powered on, the machine tool is in the G90 status.

Example:

```
N0010  G90  G92  X20  Z90  N0020  G01  X40  Z80  F100
N0020  G01  Z60  F50
N0030 G03  X60  Z50  I0  K-10
N0040  M02
```

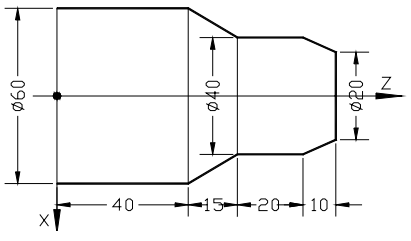
2.2.34 G91—Programming with incremental mode

Format: G91

Description: When G91 is compiled, all the coordinate values afterwards will be based on the previous coordinate position as the start point to calculate movement programming value. . In the following coordinate system, the previous point will always be regarded as the start point for programming..

Example:

```
N0010  G91  G92  X20  Z85
N0020  G01  X20  Z—10  F100
N0030  Z—20
N0040  X20  Z-15  F100
N0050  M02
```



Illus 2-26

2.2.35 G92—Setting work piece coordinate system

Format: G92 X_Z_

Description:

(1) G92 only changes the current displayed coordinate value of system, and the coordinate axes require no shift to set coordinate origin..

(2) The effect of G92 is to change the displayed tool nose coordinate into the set value..

(3) The X and Z following G92 can be compiled respectively, and can also be compiled wholly.

2.2.36 G96－Constant linear speed cutting

Format: **G96**

Description:

(1) The constant linear speed cutting only be suitable for system with analogue output, which controls infinitely variable speeds of spindle..

(2) When G96 is implemented, CNC regards the cutting speed of this time as the benchmark, and the spindle rotating speed will be regulated in linear according to whether it's feed or retracting in X direction..

(3) When spindle rotating speed reaches rated maximal rotating speed of system, the continuous feed is implemented in X direction, and the spindle rotating speed no longer changes..

(4) G96 is canceled by instructions such as G97, M05, M02 and so on..

(5) Because the spindle frequency converter has time setting of increase/decrease speed, so when G96 takes effect, the shorter the increase/decrease speed time constant, the better the following of spindle rotating speed..

(6) The lower limit of spindle rotating speed is determined by 28# parameter at constant linear speed cutting.

2.2.37 G97－Cancel constant linear speed cutting

Format: **G97**

2.2.38 G98－Cancel feed of each rotation

Format: **G98**

2.2.39 G99－Setting feed of each rotation

Format: **G99 F××××××**

Description: The F×××××× following G99 is the feed distance of each rotation, which the unit is Ųm, namely only integer can follow F. . This is different from feed of every minute.

2.3 Subsidiary function (M function)

The M function is also called subsidiary function, which is used for state control of CNC input and output. . The subsidiary function is composed of M with two digits following it, and the subsidiary functions of 31DT numerical control system are as follows:.

M00	**Program pause**
M01	**Condition pause**
M02	**Program end**
M03	**Spindle positive rotation**
M04	**Spindle reverse.**
M05	**Spindle stop.**
M08	**Open cooling fluid**
M09	**Close cooling fluid**
M10	**Work piece clamping**
M11	**Work piece loosing**
M20	**Open specified relay**
M21	**Close specified relay**
M24	**Setting cutting tool compensation number.**
M30	**Returning to program beginning**
M71~M85	**M function pulse output**

M function is the one to switch on or switch off machine tool external switches such as spindle startup, stop and refrigeration motor switching on or switching off. . M function is always different from the one prescribed by standard owing to differences of machine tool manufacturer and machine tool structure as well as model..

Then M functions will be elaborated on.

2.3.1 M00－Program pause

Format: M00

Description: M00 appears in the program, and pauses after this segment program completes. . Pressing the processing start key, and the program continues.

2.3.2 M01－Condition pause.

Format: M01 K×× or M01 L××

Description: The two digits after K (or L) is corresponding to certain number of I／O, and the program will stop here until the exterior inputs one low-valid (or high-valid) signal to the I／O, then program will continue. The valid time of external level is required greater than 100 milliseconds. . K is high-level valid and L is low-level valid. . The input port number definition of 31DT system is given below:.

1# input port———1# cutting tool
2# input port———2# cutting tool
3# input port———3# cutting tool
4# input port———4# cutting tool
5# input port———5# cutting tool
6# input port———6# cutting tool
7# input port———7# cutting tool
8# input port———8# cutting tool
9# input port———XREF
10# input port——ZRDY
11# input port——YRDY
12# input port——YPSN
13# input port———XPSN
14# input port———XRDY
15# input port———YREF
16# input port———XERR、YERR、ZERR
17# input port———+ limit
18# input port———ZREF
19# input port———emergency stop
20# input port———start
21# input port———limit
X# input port———H／L
23# input port———emergency stop
24# input port———ZPSN

Among them, 19# input port can only be used for emergency stop.

2.3.3 M02－Program completion.

Format: M02

Description:

(1) M02 completes processing program..

(2) If the SLOF digit of digit parameter 00# is equal to 0, M02 will close spindle (M05) and refrigeration (M09), and this cycle is completed..

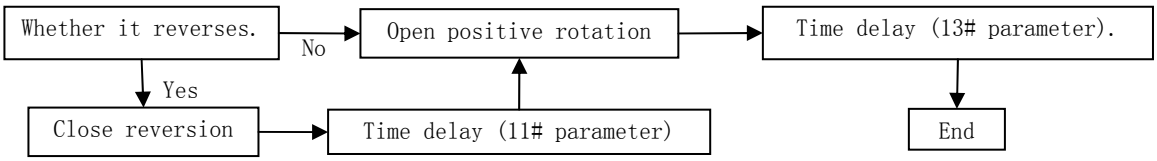
(3) If the SLOF digit of digit parameter 00# is equal to 1, M02 won't close spindle and refrigeration, and only this cycle is completed.

2.3.4 M03—Spindle positive rotation

Format: M03

Description:

(1) The M03 instruction is in the program, first the spindle positive rotation relay will be switched on, then S function outputs analogue and control spindle to rotate in clockwise direction. It controls 1# relay. . M03 movement sequence:.



(2) If 21# parameter is equal to 0, M03 will maintain output.

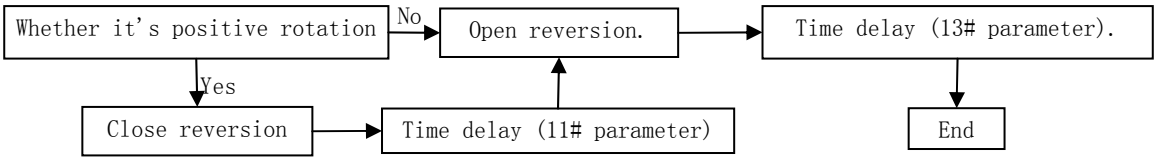
(3) If 21# parameter is equal to 1, M03 is the pulse output, and pulse delay is determined by 15# parameter

2.3.5 M04—Spindle reversion

Format: M04

Description:

(1) Controlling 2# relay and starting spindle reversion.. M04 movement sequence is:.



(2) If 21# parameter is equal to 0, M04 will maintain output.

(3) If 21# parameter is equal to 1, M04 is the pulse output, and pulse delay is determined by 15# parameter

2.3.6 M05—Spindle stop

Format: M05

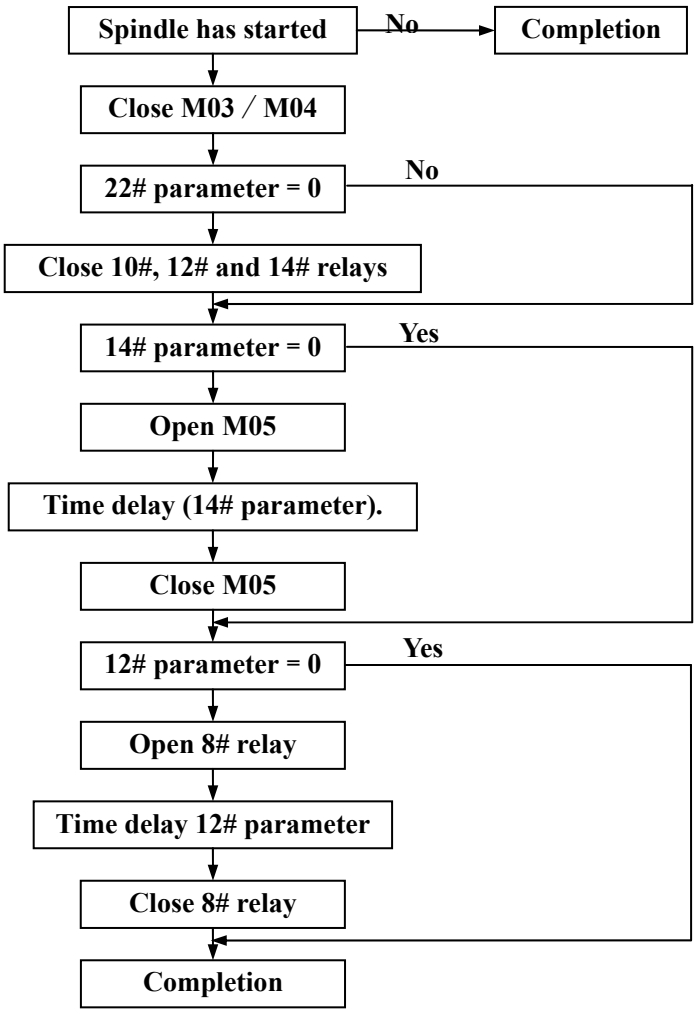
Description:

(1) M05 instruction outputs pulse signal and closes spindle positive rotation or reversion control relay to stop outputting analogue, and the spindle rotation stops. The width of output pulse signal is determined by 14# parameter. Controlling 3# relay..

(2) If 12# parameter isn't equal to 0, CNC also outputs short signal to 8# relay to offer spindle braking function..

(3) If X# parameter is equal to 0，M05 will closes 10#、12# and 14# relays; if 22# parameter is equal to 1，M05 won't close 10#、12#、14# relays。

(4) The execution course of M05 instruction is:.



2.3.7 M08—Open cooling fluid

Format: **M08**

Description: M08 function will be executed at the beginning of this segment program, and the cooling fluid control relay will be switched on (6# relay).

2.3.8 M09—Close cooling fluid

Format: **M09**

Description: M09 function will close cooling fluid control relay (6# relay) after this segment program completes.

2.3.9 M10—Work piece clamping

Format: **M10**

Description: M10 function will switch on the relay controlling clamping movement (9# relay).

2.3.10 M11—Work piece loosing

Format: **M11**

Description: M11 cancels relay controlling clamping movement (9# relay)

2.3.11 M20－Open specified relay

Format: **M20 K××**

Description: The two digits after K is corresponding to certain relay number, and output of OC port will be valid when executing to this statement, which switches on the relay on the relay panel and continues after delaying time for one second.

2.3.12 M21－Close specified relay

Format: **M21 K××**

Description: The two digits after K is corresponding to certain relay number, and output of OC port will be invalid when executing to this statement, which switches off the relay on the relay panel and continues after delaying time for one second.

2.3.13 M24－Setting cutting tool compensation number

Format: **M24K××**

Description: When the cutting tool compensation number is used for the first time and must be changed, it only changes cutting tool compensation number instead of regulating work piece coordinate, and the two digits after K is from 00 to 10.

2.3.14 M30－Returning to program header

Format: **M30**

Description: When program executes to M30, the system will points the program pointer to the program beginning and waits for input of operator; if start key is pressed, the program will be executed again and if the cycle key is canceled, the processing will be canceled.

2.3.15 M71~M85－M function pulse output

Format: **M77／（taking M77 for example）**

Description: Considering that various machine tools have different requirements to M function, the system sets up this function which is used to control relay on the relay board to output short time on-off signals, and its movement sequence is as follows (taking M77 for example):.

(1) Controlling 7# relay on the relay board and switching on it..

(2) Time delay 15# parameter. When 15# parameter is equal to 0, the time delay will be 0.4 second..

(3) Switching off 7# relay.

2.3.16 Output interface, programming and relay number comparison table

Socket and port number	Pulse output	Maintain output	Cancel output	Supplement function	Remark
8J1—P2(P6)	M71	M20 K1	M21 K1	M03	M03，M04 interlock
8J1—P3(P8)	M72	M20 K2	M21 K2	M04	M03，M04 interlock
8J1—P4	M73	M20 K3	M21 K3	M05	M05 cancel M03、M04
5J1—P6	M75	M20 K4	M21 K4	Holder positive rotation	
5J1—P7	M74	M20 K5	M21 K5	Holder reversion	
5J2—P5	M76	M20 K6	M21 K6	M08	M09 cancel M08
5J2—P12	M77	M20 K7	M21 K7	M12／S3／M16	S1、S2、S3 interlock
5J2—P4	M78	M20 K8	M21 K8	Braking	

5J2--P11	M79	M20 K9	M21 K9	M10	M11 cancel M10
4J3—P3	M80	M20 K10	M21 K10		
4J2—P2(P10)	M81	M20 K11	M21 K11		
4J2—P3	M82	M20 K12	M21 K12		
4J2-P2（P10）	M83	M20 K13	M21 K13		
4J1—P3	M84	M20 K14	M21 K14		
4J1—P2（P10）	M85	M20 K15	M21 K15		
5J2—P3		M20 K16	M21 K16	M15／S2	S1、S2、S3 interlock
5J2—P10		M20 K17	M21 K17	M14／S1	S1、S2、S3 interlock
4J1—P2（P10）	M85	M20 K15	M21 K15		

2.4 F、S、T functions

F, S and T functions are the abbreviation of feed function, spindle function and cutting tool function.

2.4.1 F—Feed function

The feed function is called F function in general, and F function can directly provide feed speeds of G01, G02 and G03. F function is signified by F and following digits, which the feed speed is mm/min. . The feed speed of 31DT numerical control system is between F1 mm/min to F 60000 mm/min, and user can randomly select according to actual cutting condition.

2.4.2 S—Spindle speed control

The spindle control is divided into the frequency control motor and spindle-use change speed motor (two-speed motor and three-speed motor).

2.4.2.1 The spindle with frequency conversion motor

The spindle control is implemented by the S××××, M03, M04, M05, 03# parameter, 04# parameter and 11# ~ 14# parameters which determines the analogue output controlled by the spindle (the factory set is 0-10 V).. In general, the machine tool spindle has one level manual gear shift to implement various rotating speed scope, which the bigger output torque still exist at low speed.. 31DT can output high speed and low speed analog voltage, and the top limit of high and low gear is determined by 3# parameter and 4# parameter in the P parameter..

For example: 03# parameter = 5000，04# parameter =1500

When it's at the high gear (the 'H/L input' must be switched off with the '24V earth'): Vs = (Vref*s) 03# parameter.

Vref: Maximal analog voltage 10V;

Vs: The analog voltage actually output by CNC;

03# parameter: High gear top limit;

04# parameter: Low gear top limit;

When it's at the low gear (the 'H/L input' must be switched on with the '24V earth'): Vs=(Vref*s) 04# parameter.

Vref: Maximal analog voltage 10V;

Vs: The analog voltage actually output by CNC;

03# parameter: High gear top limit;

04# parameter: Low gear top limit.

2.4.2.2 The spindle operation controlled by digital quantity

The instructions are S1, S2 and S3 (suitable for multi-speed motor), and used with M03, M04 and M05.. M03, M04 and M05 determine the positive rotation of spindle.

2.4.3 T—Cutting tool function

The cutting tool function is also called T function used to select cutting tools, and the use of electro-holder or tool array are determined by 05# parameter (=0: electro-holder, =1tool array).. The cutting tool function is signified by T and following digital block. Towards the electro-holder, the revolving holder is controlled to implement tool-change and change corresponding cutting tool number; and towards tool array, the numbers are all 0, only the cutting tool compensation number is changed.

2.4.3.1 T function format.

Tn m

n: cutting tool number(1—8)

m: cutting tool compensation number (1—10)

There are several following written forms:

	Electro-holder tool array	
Tn	**Changing No. n cutting tool and using No. n cutting tool compensation**	**Using No.n cutting tool compensation.**
Tn.0 or Tn	**Changing No. n cutting tool without cutting tool compensation**	**As above**
T0.m or T.m	**Not changing cutting tool and using No. m cutting tool compensation**	**Using No.m cutting tool compensation**
Tn.m	**Not changing cutting tool and using No. m cutting tool compensation**	**Using No.m cutting tool compensation**
T0.0	**No movement**	**No movement**

2.4.3.2 The cutting tool number and cutting tool compensation number

Towards the tool array, the tool numbers are all 0, and the deviation of tool nose will be modified through changing tool number; towards the electro-holder, the tool numbers are acquired by sensor in the holder, and they are not memorized by CNC. . Accordingly, no matte whether it's tool array or electro-holder, CNC can correctly learn about the tool number. However, the cutting tool compensation number can't be acquired through external switch, especially towards tool array and when the tool number is different from cutting tool compensation number, CNC can only acquire the cutting tool compensation number via memory. In normal work, the cutting tool compensation number is between 1 and 10, but under special condition, such as CNC is used for the first time, the system is totally cleared, or the cutting tool compensation number exceeds the scope above caused by disordered memory or it's not exact, the cutting tool compensation number can be specified artificially with M24, but at this time the CNC actual coordinate is probably different from displayed coordinate, and the re-centering CNC coordinate is required..

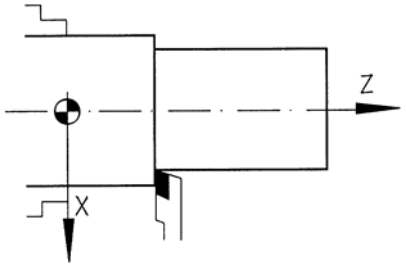
Note: If the parameter is set for electro-holder (05# parameter = 0), while CNC system isn't connected with the electro-holder or is wrongly connected, then the system will appear error 06.

2.4.3.3 Cutting tool compensation table

The physical meaning of cutting tool compensation table is the difference of tool nose among various tools. In order to decrease memory capacitance, the general cutting tool compensation table only memorizes coordinates in X and Z directions relative to one fixed benchmark of every tool, then at the time of cutting tool compensation, according to different difference of the current tool and the previous one relative to this fixed benchmark, which the offset between the two tools can be acquired, that is the cutting tool compensation table doesn't memorize the cutting tool compensation value, and the cutting tool compensation value can be acquired through calculation at the time of tool-change. . 31DT has 10 cutting tool parameter addresses in total, namely from T01-T10.. Under PARAM status, 10 cutting tool parameters are displayed in the screen by pressing n key, and DX is the cutting tool compensation value in X direction, and DZ is the cutting tool compensation value in Z direction. The operator can change the actual cutting tool parameters in this screen.

2.4.3.4 Confirmation of the work piece coordinate system

Once the cutting tool compensation table is confirmed, the tool position of current tool must be confirmed to the coordinate value in the work piece coordinate system, now the origin of work piece coordinate system is supposed on the chuck (like the right figure below):.



(1) Using one excircle tool of 90 degree to cut one section of excircle..

(2) Measuring the diameter a. b (mm) of cut excircle..

(3) Using G92 to input Xa.b and pressing 'ENTER' key. Among them, a. b is the outside diameter of actual measurement..

(4) Retracting cutting tool to proper position and re-cutting end surface..

(5) Using measuring gauge to measure the distance c. d (mm) from chuck end surface to the work piece..

(6) Using G92 to set Zc.d and pressing 'ENTER'. The Z coordinate of tool nose relative to this work piece coordinate system is acquired..

(7) If the work piece end surface is as the coordinate origin, then the Zc.d should be changed into Z0.

2.4.3.5 About initialization of cutting tool compensation number

When CNC is used for the first time, the cutting tool compensation number can be generated with the following methods:.

(1) (This method is recommended).

① Using Tn instruction to change tool and adopting this cutting tool compensation number.

② Measuring the tool nose position (the method of using this tool for one section of trial cut), and the X and Z actual coordinate values will be set with G92.

(2) Adopting M24 Km to set one cutting tool compensation number artificially.

Once the cutting tool compensation number is generated, CNC will continue to memorize, and this memory is saved in the memory protected by battery, but for a variety of reasons, the cutting tool compensation number will be probably damaged. In order to prevent this condition, it's recommended to adopt mnemonic function to save crucial data in CNC into the electronic

disk before shutdown, and then to recover the saved data after every starting up..

Steps:

Shutdown save:

A) The "PARAM parameter" primary function can be accessed.

B) Pressing "◄►" and then pressing "F1 (shutdown)".

C) Shutdown.

Starting up recovery.

A) The "PARAM parameter" can be immediately accessed after starting up.

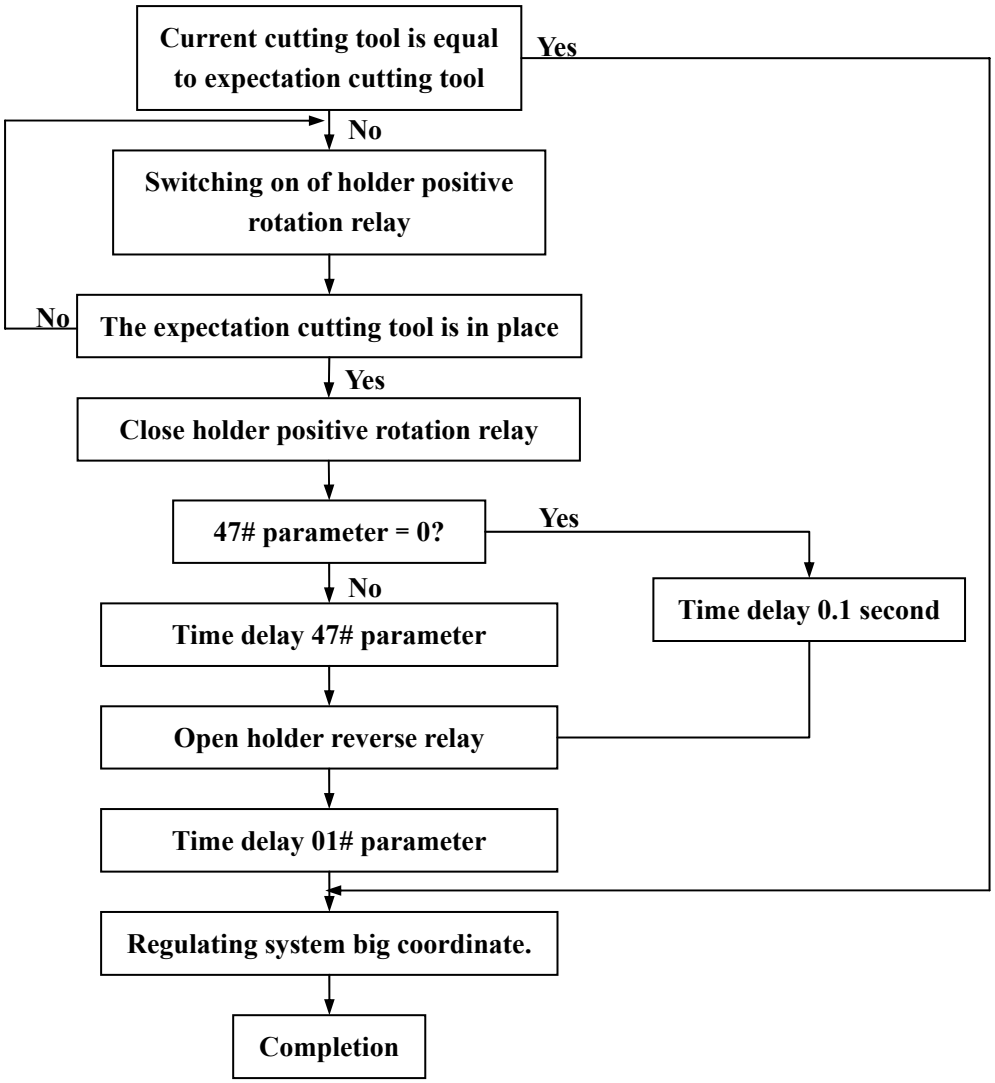
B) Pressing "◄►" and then pressing "F2 (starting up)".

C) Quit.

The methods above can protect and recover the following data: cutting tool compensation number, system coordinate and clearance compensation direction.

2.4.3.6 T function movement sequence

The CNC built-in PLE movement sequence when T function leaves factory is as follows:.



2.4.3.7 Presetting cutter methods and steps

This system adopts universal presetting cutter method of trial cut, and for the convenience

of operation the system offers independent memory presetting cutter mode in X and Z directions, and the presetting cutter in X and Z directions is memorized simultaneously, the specific operation steps are as follows:.

(1) Independent memory presetting cutter in X and Z directions.

Presetting cutter method in X direction:

①. One work blank piece is clamped on the chuck;②. Accessing manual operation mode;③. Turning the holder and selecting the cutting tool number requiring presetting cutter, such as "T1";④. Setting appropriate spindle rotating speed and manual feed speed, and starting spindle, moving holder, using selected No.T1 cutting tool to turn one section of excircle (or inner bore), and "X memory" appears at the top right corner in the screen by pressing "Xsav" key; ⑤.Manually quitting cutting tool and stopping the spindle, measuring the diameter of excircle (inner bore) after cutting;⑥ Pressing "cutting tool compensation setup" key to access the cutter compensation parameter interface, and "X 0.000" appears in the screen, inputting the diameter measured in the step 5 on the keyboard, and pressing "ENTER" key for confirmation then pressing "Save Open program" for saving..

Note: If the cutting tool cuts on the other side of the work piece axial line (namely the cutting tool is in the opposite direction of axial line), then the input diameter is negative..

Presetting method in Z direction.

①. Accessing manual operation mode;.

②. Re-starting the spindle and moving holder, and using No. n cutting tool to turn one end surface on the work blank. "Z memory" will appear at the top right corner in the screen by pressing "Zsav" key;.

③. Manually quitting the cutting tool and stopping the spindle, measuring the length L from the cutting end surface to the chuck end surface;.

④. Pressing "cutting tool compensation setup" key to access the cutter compensation parameter interface, and "Z0.000" will appear in the screen. Inputting the length L measured in the step ③ on the keyboard and pressing "ENTER" for confirmation, then pressing "Save Open program" for saving. At this time, the relative position of this cutting tool in the work piece coordinate system has been confirmed..

Note: At the time of presetting cutter for every tool, the measurement benchmark must be the same; else the cutting tool compensation value in Z direction will be incorrect..

(2) The presetting cutter is memorized in X and Z directions simultaneously.

①. One work blank piece is clamped on the chuck: ②. Accessing manual operation mode;③. Turning the holder and selecting the cutting tool number requiring presetting cutter, such as "T1";④.Setting appropriate spindle rotating speed and manual feed speed, and starting spindle, moving holder, using selected No.T1 cutting tool to turn one end surface on the work blank, and pressing "Zsav" key to quit the cutting tool to proper position along X direction, to process one section of excircle, pressing "Xsav" key. ⑤.Quitting the cutting tool and stopping the spindle, measuring the diameter of excircle (inner bore) after cutting and the length L from the work piece end surface to the chuck end surface; ⑥ Pressing "cutting tool compensation setup" key to access the cutter compensation parameter interface, and "X 0.000" appears in the screen, after the diameter is input, pressing "ENTER" key for confirmation. At this time, the cursor stops after the X direction offset and flashes, pressing "F3 (cutting tool compensation)" and "Z 0.000" will appear in the screen. After the length L is input, pressing

"Save Open program" key for saving. After the cutting tool compensation is established, the relative position of this cutting tool in the work piece will be determined..

Description: At the first time of presetting cutter after clearing memory, it's necessary to press "F3 (cutting tool compensation)" after accessing the cutter compensation parameter interface, then the system will appear "X 0.000" or "Z 0.000", at this time the cutting tool compensation values in X and Z directions can be input. . Pressing "ENTER" for confirmation after the input completes.

2.4.3.8 Directly inputting cutting tool compensation value

If it's necessary to directly input the cutting tool compensation value of certain number, pressing "cutting tool compensation setup" under manual status to access the cutter compensation parameter interface, and moving the cursor in front of the corresponding cutting tool compensation value, pressing "F5" to directly input the measured cutting tool compensation value, pressing "ENTER" for confirmation and then pressing "Save Open program" for saving.

2.4.3.9 The minute modification and regulation of cutting tool compensation value

In the actual processing, if the dimension of work piece processed by certain cutting tool is bigger or smaller, the method of changing cutting tool compensation value for compensation; if the X direction is greater than 0.05 mm, the cursor will be moved to the cutting tool compensation parameter position of this cutting tool number, and pressing "F4 (modification and regulation)" then pressing deviation value 0.05. . Then pressing "Save Open program". . Inputting positive value if it's bigger, and inputting negative value if it's smaller. . The modification and regulation of Z direction is as the same.

2.4.4 Axis optional function

Under machine tool zero or hand wheel status, pressing "F4(X Z)" key to select effective axes, at this time, the system will display the current effective axes in the operating status bar as the X axis (or Z axis).

2.4.5 Coordinate modification and regulation function

Under manual, automatic or pulse control status, pressing "PARAM (parameter)" key to access the parameter interface, and pressing "F4 (modification and regulation) pressing" to access the coordinate modification and regulation interface after pressing "◄►" key to switch to the second page of the menu, inputting modification and regulation value in X direction, after pressing "ENTER" for confirmation then inputting modification and regulation value in Z direction. . After the modification and regulation of coordinate is completed, the coordinate value (including big coordinate and small coordinate) displayed by the system immediately changes..

Note: If all dimensions processed are bigger, inputting positive value. . If they are smaller, inputting negative value

Chapter Three System Operation

Properly operating 31DT must master the operating methods and displayed meanings of all sorts of information of various functions. The operational interface which is offered by numerical control system to users is as follows:.

(1) Keyboard panel: Receiving user's instructions to system, and accordingly coordinating system internal status to implement entire system functions.

(2) Communication interface: The communication can be implemented with any computer equipped with standard RS 232 serial interface..

(3) 6 Inch LCD, which can offer various real-time system information.

(4) Various input/out interface.

3.1 Safety, protection and compensation

In general situation, the ring opening driving of stepping machine won't influence much on the machinery at the time of overtravel blocking rotation owing to its principle, while towards the system which the exchange servo is as the executing component, the overload capacity of exchange servo and output torque will increase dramatically, and the mechanical damage even serious accident will probably occur. So the safety protection of machine tool is significantly important towards the machine tool driven by exchange servo-unit. The system limits probability of error occurring through various aspects below.

3.1.1 Emergency stop

The emergency button should be equipped with a pair of normally opened/normal closed contact for each, and the normally opened contact should be connected to P6 and P9 of system 5J3 (see technical manual)..

It's strongly recommended that the normally closed contact is switched in the strong power circuit of machine tool, which is the control circuit supplying power for main circuit (spindle and servo), so as to guaranty that the spindle and servo can stop operating with the maximal reliability under emergency..

When the emergency stop signal is received by the system, it'll be switched to the manual mode and 55# alarm occurs, and all operations are locked.

3.1.2 Hard limit

Towards the machine tool which the exchange servo is as the executing component, the mechanical type two-way or three-way travel switch should be equipped, which can cut off the power supply control circuit of main circuit forcedly when the soft limit doesn't work (see emergency stop). Towards general three-way switch: One-way is switched in the strong power control circuit, one-way is switched in the limit input of system, and the second way is as the coarse positioning signal of returning to machine tool zero..

Owing to that the action of proximity switch can't directly cut off the control circuit, so it's not recommended to use inductive type proximity switch as the limit switch, if it must be used, the proximity switch with OC port output of NPN type should be selected.

3.1.3 Soft limit

The system offers internal timing examination function to implement real-time monitoring of system coordinate whether it exceeds the interval set artificially, once it's exceeded, the operation should be stopped and switched to manual mode with 40# alarm; this course is composed of a series of reference system..

(1) Whether the machine tool coordinate (SCOR=1)(reference point) or the working coordinate (SCOR=0, big coordinate) is as the benchmark of soft limit coordinate is determined by the SCOR digit of 09# digit parameter..

(2) Whether the soft limit function is valid after returning to machine tool reference point (SNZ=0) or needn't returning to reference point (SNZ=1) is determined by the SNZ digit of 02#..

(3) The interval set by soft limit is defined by the system parameter (P parameter) 60#---65#, once the coordinate (machine tool coordinate or work piece coordinate) selected by the system exceeds interval of each axis, the system will give alarm (see parameter table)..

(4) When the limit occurs, whether each axis stops suddenly or stops with speed reduction is determined by OVS of 09# towards hard limit, and determined by the SLS digit of 02# towards to soft limit..

(5) When the soft/hard limit stops with speed reduction, its time constant of negative acceleration is determined by the 44# parameter, while the maximal speeds are all set as 10 m/min..

(6) When limit sudden stop is selected, the servo will generate great impact and generate adverse effect to the machine tool, so adopting stop with speed reduction is recommended, but over-impact will be caused and the soft limit region is exceeded; the smaller its acceleration is, the longer the interval-exceeding is, so the method of using time constant (44#) to increase acceleration and decrease border crossing length.

3.1.4 Clearance compensation

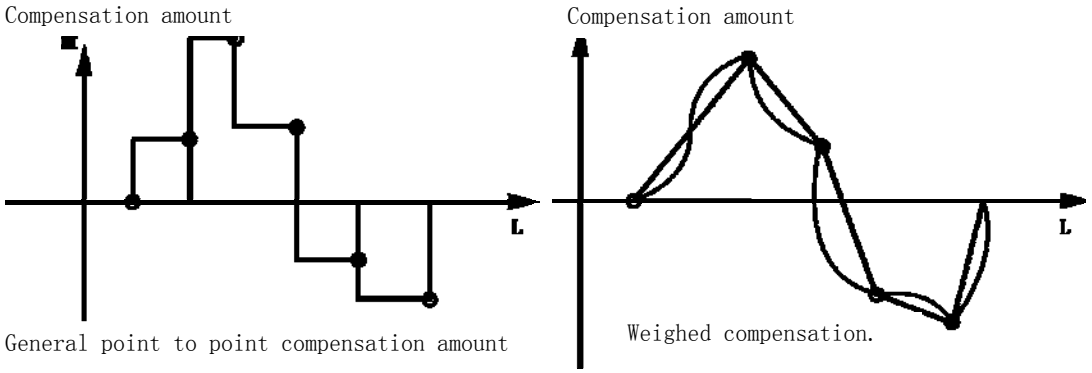
Towards the mechanical transmission machinery with certain reversion clearance compensation, the system can compensate the precision loss but it can't be expected that the effect after compensation is the same as the gapless processing quality, especially when the arc processing exceeds quadrant, the bigger the clearance value is, the more influence on the quality, so the machine tool should decrease the clearance compensation value as far as possible..

The system adopts principle of extra-movement to deal with clearance, and its acceleration of movement is calculated by the 39# time constant and 48# compensation speed top limit.

3.1.5 Screw thread pitch compensation

Owing to influence of composite factors such as manufacture and temperature, the screw thread pitch error statistically belongs to the system error and not the accidental error, and the numerical control system thinks that when the measurement distance of error between two adjacent measuring points is small enough (≤1.5-2 times of thread pitch) , the error presents weighted linear distribution. When the system implements thread pitch error compensation, it not only guaranties the accurate compensation of the measuring point, but also the weighted compensation can be implemented to the errors among measuring points, so as to guaranty that every system cycle (about 4ms) can implement compensation to the screw error within the total

travel of entire screw instead of only compensating the measuring point independently

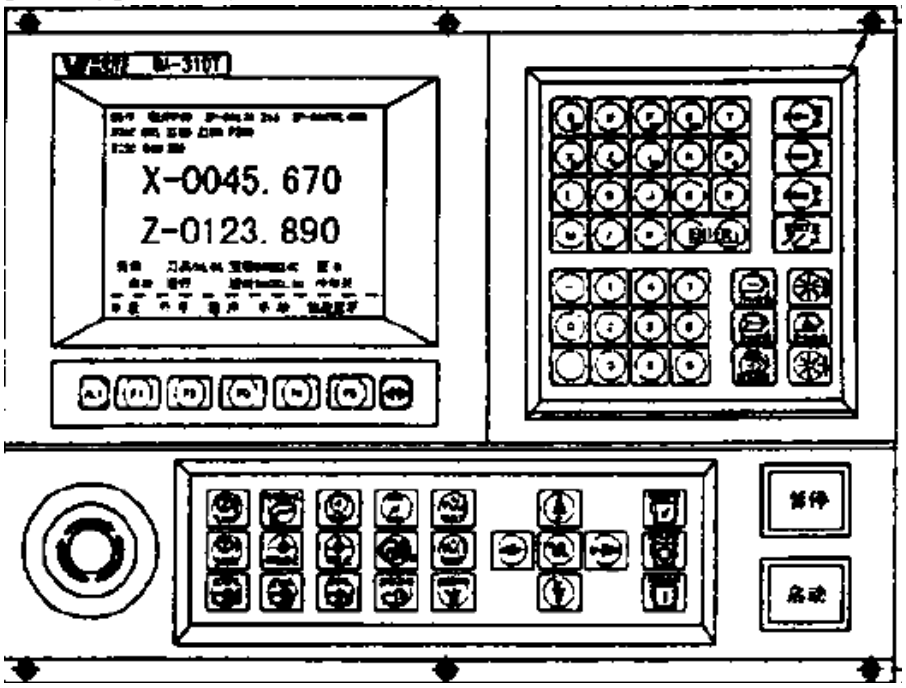


Seen from the weighted compensation figure, the compensation amount of any point of the screw length L is different from the adjacent measuring point, while the compensation amount is not only related with the error value of two measuring points at both sides of current point, but also related with the fore-and-aft error change rule of adjacent measuring point. Seeing 4.5.1 for the implementation of thread pitch compensation.

3.2 Operating keyboard of numerical control system

The whole operating functions offered by numerical control system can be implemented by keyboard operation.. The front panel of 31DT system is composed of 6 Inch LCD, address function keyboard region, numeric keyboard region and manual operation keyboard region..

The operating panel of system is as the figure below:.



31DT system has three major functions.

PRGRM-Program: Various management, program input and output operation related with program..

OPERT——Process: All movements and strong signal control related with machine tool..

PARAM——Parameter: The control parameters are set according to various machine tools.

3.2.1 Primary function key

(1) PRGRM (Program): The user processing program management which is used to manage the processing program compiled by the user, and under this primary function there are sub-functions related with program management..

(2) OPERT (Process): It's the machine tool operating, towards various operating functions, they can be implemented in the sub-functions under this function..

(3) PARAM (Parameter): It's the parameter setting which is used to set various parameters related with machine tool or numerical control system.

3.2.2 Edit character key.

It's primarily used to input ISO code and parameter values of various coordinates of processing program..

G M F S T X Z P I K R N L D J

0 1 2 3 4 5 6 7 8 9 are numetric keys

/ **Tilted number key.**

␣ **Space key.**

— **Minus key.**

。 **Decimal key.**

EBTER **Is the enter key, which moves cursor to the head of next line when editting, while in other status, it indicates that the input ends here..**

= **Equal sign key for parameter assignment**

3.2.3 Coordinate feed and feed parameter setting of manually operating machine tool

The symbols of ◄Z-, ►Z+, X-▲ and X+▼ respectively signify manual feed along Z negative direction, Z positive direction, Xnegative direction and X positive direction..

Gmdi Fset Iset and Sset are respectively used to access MDI mode at manual feed, and set feed speed F, stepping amount I and spindle rotating speed S..

When Xsav (Zsav) measures the cutting tool compensation value, it memorizes the coordinate value of X (Z) direction.

3.2.4 Soft definition key F1~ F5

There are five keys marking F1~F5 in the lower part of the 6 Inch LCD, the function of the key will change with the primary function selceted by current user, which is primarily used to select sub-function belonged to the primary function under it. The meaning of F key is corresponding to the Chinese character in the lower part of current screen.. When exceeding 5 keys, pressing "◄►" key to switch to soft function menu of the next page.

3.2.5 Others

Cycle startup, which is used to execute one processing program.

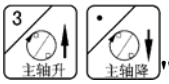
Cycle pause.

Cycle cancel, termination of this cycle.

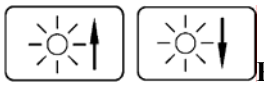
Manual fast: Simultaneously pressing either key of the four keys of ◄Z-, Z+►, X-▲ and X+▼, the machine tool will operate with the manual maximal speed set by the 10# parameter..

Hand wheel pulse multiply factor setting: The output pulse of hand wheel is multiplied by the set multiply factor.. ×1、×10、×100

Feed multiply factor increase/decrease: Dynamically regulating feed speed F under automatical and manual modes..



"Spindle decrease" Spindle multiply factor increase/decrease: Dynamically regulating spindle rotating speed S under automatical and manual modes (only valid for main motor speed control by frequency variation).


Regulating brightness of LCD

This key is the shortcut one, under status of manual, automatical and hand wheel operating; the cutting tool compensation parameter interface can be fast accessed by pressing this key..

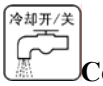
Save key of file call: After inputting program name, the program to be processed currently will be called, and the program, system parameter, cutting tool parameter and files of machine tool parameters will be saved in the electronic disk..

Single-step tool-change: The tool will be changed to the next in turn by the system for each pressing..

Spindle positive, clockwise rotation.

Spindle reversion, counterclockwise rotation.

Spindle stop.



Cooling fluid open/close, switched by pressing.

Return to prsetting cutter point: It's equivalent to G76, and at the begining off the work piece, useing 18# and 19# to set one coordinate value as the processing start point, after processing completes every time, the holder return to this point, which can implement measurement and clamp work piece again and so on..

"Return to reference point" fast G74 (shortcut key of function G74 X, Z).

"Reset" Computer hardware system resets, and accesses initial status after starting up.

"CAN" CAN (Cancel), cancel the displayed error code.

"ALT" When certain parameter (such as P parameter, digit parameter and thread pitch compensation parameter) can't be fully displayed in one screen, pressing "ALT" key to switch among screens..

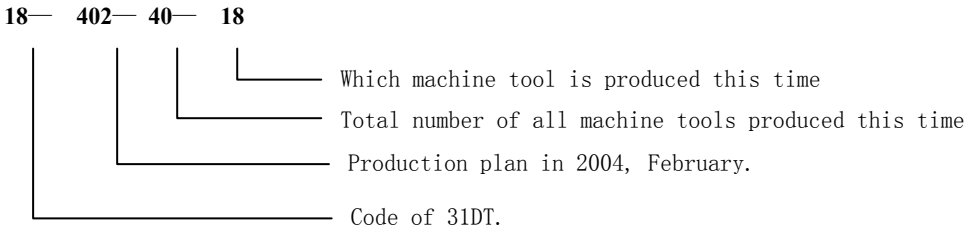
Note:The system keyboard has several diplex keys, and 31DT CNC system can automatically judge the meaning of keys; the user requires no switching operation of key definition.

3.3 Starting up

Before the first starting up, the system appearance should be examined to see whether there is obviously abnormal condition, whether the power connection is not right, whether the joint to the switch power comes off, and it can be powered on after confirmation. . The system power source is the three core power plug, which uses single-phase 220V/50HZ A.C. with grounding.

3.3.1 Starting up, image and design

When the system starts up, it displays the image of starting up, in which there are software version series and version number such as DT-V 3.2, another line is the output serial number of this machine tool, for instance:


18— 402— 40— 18
Which machine tool is produced this time
Total number of all machine tools produced this time
Production plan in 2004, February.
Code of 31DT.

The serial number of each machine tool system is different, if the same serial number is found, please contact with the corporation for fear that it's pirate..

The logo of system can be designed by user himself, and user can design one piece of image with dot matrix of 320x240, and downloads it into the system through serial communications mode, the specific processes are as follows:.

(1) Using input function in the PRGRM primary function to set the filename as 000, and the imported file is the image file of starting up..

(2) Utilizing the special image transfer soft ware of our corporation to transfer the designed image.

(3) If you don't want to display software version or serial number, please set the 17# system parameter as 1.

(4) When 17# is 0, the user should check whether serial number is repeated.

(5) Please consult the corporation for related detailed process.

3.3.2 Selection of primary function

After starting up, only the primary function selection is valid, if user wants to use some one specific function, it can be implemented only after accessing relevant primary function.. Accordingly, the user must press the selection key of primary function which includes the specific function (subfunction) first of all, and selects desirable function after accessing corresponding primary function status.The operating interface of this system adopts note mode as far as possible, which makes user learn about whether the current operation is offered by the system.. Simultaneously, the principle of key-press operation is: Implementing user's desirable functions with fewest key-press numbers.

Gengerally, in order to implement some one specific function operation, the user simply presses the key three times.

(1) Pressing the primary function key, the system will quit original status and and access new primary function status.

(2) Pressing sub-function selection key (it's one of F1～F5 in general), and setting subfunction valid..

(3) Under subfunction status, pressing specific operating key to implement specific function.. For instance, the current system is in the editing mode and is editing processing program, the user hopes to access operating status and moves X and Z axes to proper position, and the operation is as the following sequence:.

① Pressing OPERT (Process) primary function key, the system will automatically save the editted processing program, then quit "PRGRM" primary function and access "OPERT" primary function, the main image of "OPERT" is displayed in the screen.

② Pressing coordinate moving keys◄Z-、Z+►、X-▲、X+▼ to move coordinate axis。

This is one general operation, 2 to 4 times of operation are probably required towards other conditions.

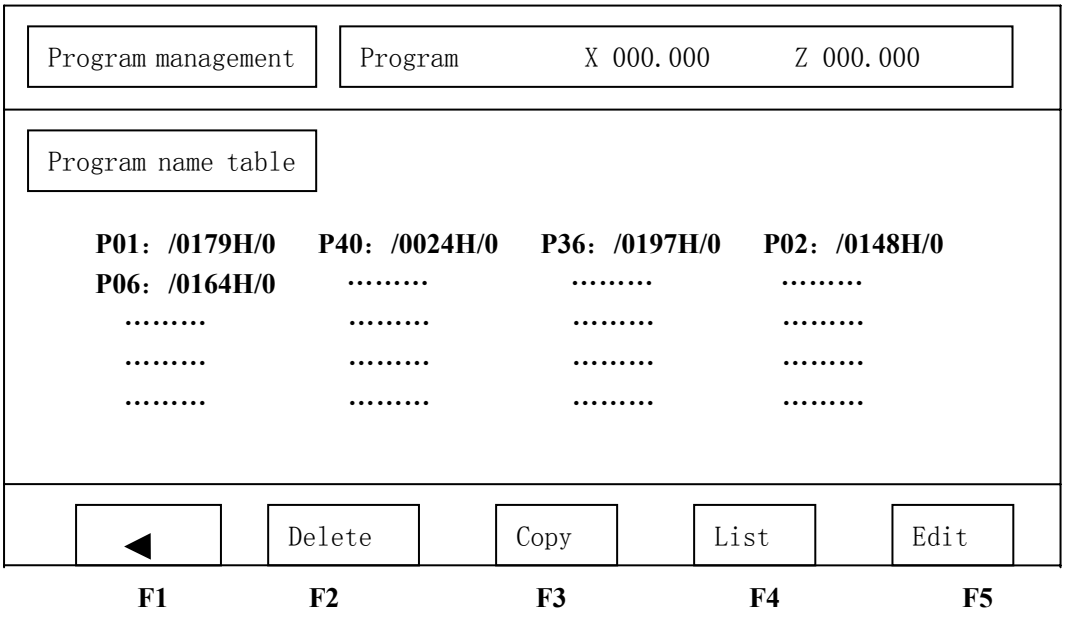
3.3.3 Selection of sub-function

There are 5 keys marking F1～F5 in the lower part of screen, the roles of these five keys are to select various sub-functions under some one primary function.. Owing to that primary function has three kinds while various sub-functions under primary function are also different, accordingly the roles of F1~F5 change at any moment, and towards to current specific

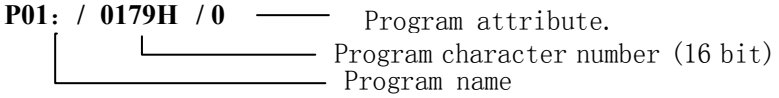
definition of F1~F5, the notes are displayed in the screen. Accordingly, F1~F5 are also called soft definition keys or F function keys. Towards to the undefined F key under current primary function, there are no corresponding notes in the screen in general, and the system makes no response with pressing this key. This system will probably define it when the software version upgrades. When F1~F5 are not enough, the key "◄►" can be pressed to switch to the next page of F function key.

3.4 PRGRM (Program) primary function

PRGRM（Program）is the user processing program management, after pressing "PRGRM" (Program) key, the main image of "PRGRM" (Program) is displayed in the screen such as the figure below.



The system offers 30 program directories, and user can most save 30 processing programs in the system, in addition, the program still remains in CNC after shutdown or power down.. Towards every program, the system displays:



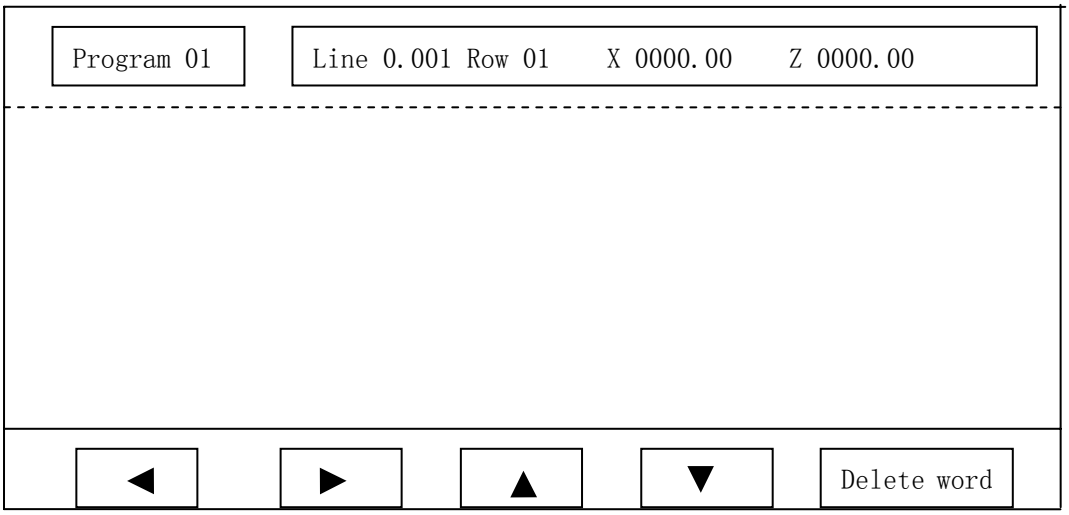
3.4.1 Input principle of program name

In the system, only the main porgram can process, and the main porgram regards P as the first letter.. The sub-program regards N as the first letter and it can only called by main porgram.. The main program or sub-program following two digits signifies various program numbers, and 8T system provides that the main program can be any one from N00 to N99, and sub-program can be any one from N00 to N99.

When inputting program name, first typing in P (or N), then inputting two digits by pressing keys from 0 to 9, if 27# parameter is equal to 0 after pressing enter key, the system won't automatically generate block number, and when 27# parameter is not equal to 0, the block number will be automatically generated.. The block number increment is 27# parameter, and system will process the input program name when input completes.

3.4.2 Program edit.

Pressing F5 in the program management image, which the word "edit" appears in the right of the third line, simultaneously the cursor flashes after the "Program name" in the first line. User can input one main porgram name P00～P99 or sub-program name N00-N99, and access edit page after pressing enter key.. The edit page is as follows:.



If this program has existed in the system, then it will be displayed in the screen. If it's the new program, there is no program in the program display region; the blank in the middle of screen is for user to edit.

The current program name as well as line number and row number are displayed in the first line; X and Z are the coordinate values which are relative to program origin of two machine tool coordinate axes. The bottom line is the note of edit function, and F function key becomes cursor movement key when editing program by this time. . "◄►" is used to switch cursor function key. The program display region is in the middle, and user can edit program in it..

Character key and numerical key: The pressed character and number will appear in the cursor position with pressing character key or numerical key, simultaneously cursor and following character all move backward one character position, namely one character inserts in the cursor position (insert mode)..

Edit function key: It's composed of F function key and enter key ENTER. The role of edit function key is to move cursor to proper position, so as to add, delete one (or one line) character and they don't directly add character in the program. . The role of enter key is to move cursor to the head of next line. If it's in the certain line of program, it will insert one blank line between this line and the next one, and simultaneously bring all characters after cursor position of original line to the blank and generate one new line. . " ◄► " can be used to switch functions of F1~F5 keys。

For instance: Edit the two segments of programs below:.

N0010 G00 X100

N0020 G01 Z10

The sequence of key-press is：N—0—0—1—0—G—0—0—X—1—0—0—ENTER

N—0—0—2—0—G—0—1—Z—1—0—ENTER

The following keys are valid when editing:

F1: Cursor moves to the left for one character and the program content changes none. . It's invalid to press F1 when cursor reaches the line head.

F2: Cursor moves to the right for one character and the program content changes none. . It's invalid to press F2 when cursor reaches line end.

F3: Cursor moves upward one line, if this line is the first one when cursor reaches it of program region in the screen, it will be invalid to press F3. . If it's not the first line of program, the programs in the full screen move downward one line with pressing F3.

F4: Cursor moves downward one line, if this line is the last one when cursor reaches it of program region in the screen, it will be invalid to press F4.. If it's not the last one of program, then the programs in the screen move upward one line.

F5: Deleting one character before cursor, and simultaneously all the characters of this line after this character move forward one character position, so as to fill up vacant position. Pressing F5 will move this line to the end of the previous line when cursor is in the line head.

After pressing "◄► "key:

F3: The full line will be deleted at the cursor position, and simultaneously the program below the deleted line moves upward one line to fill up the vacant position.

F4: Going to the previous page, and the screen can display 13-line programs which is call one page. The role of P4 is to make the last line program displayed in the current screen as the first line of the next page and displays it in the screen again. Namely all program lines after the original last program line move upward 12 lines in turn.

F5: Skipping to the next page, which is used to move the first line in the current screen and its previous program lines downward, accordingly the original first line becomes the 13th line in the screen, namely the programs move forward 12 lines.

The primary function of F4 and F5 is to move cursor to the user's required program line rapidly.

Note:

(1) If any character is not input or all the characters in the old program are deleted when editing new program, this program won't be saved which is equal to be deleted.

(2) Abnormally quitting edit mode is strictly prohibited, else part of the program even the whole program will be corrupted, these abnormal program-quitting conditions include:

A) Pressing reset key when editing, shutdown or instant power down of power grid.

B) If one large-size program is edited, the system will probably access edit page after waiting for a moment. Part of or whole program will be corrupted if the conditions above occur when waiting.

If this condition occurs, this system won't guaranty completeness of program.

If you want to quit edit mode, pressing certain primary function key and the system will be switched to other primary function status. The system will automatically save the user's edited program after processing before quitting, and the program will be recorded in the directory which is reflected in the program tab.

3.4.3 Copy, delete and program status

The three sub-functions under PRGRM are principally used for changing the program integral status.

3.4.3.1 Copy (F3)

It will copy certain program to another program. Pressing "PRGRM (program)" to access program management, then pressing "F3" to display 'Copy', and simultaneously cursor appears after the first line 'program'. Typing in the copied program name in the cursor position such as P67 (P67 must be the program which has been in the system). One 'arrow' appears after this program name after pressing enter key, and cursor flashes after this arrow, then inputting the copied program name such as P68 (P68 must be the program which is not in the system). The copy completes after pressing "ENTER" and new program generates, simultaneously corresponding changes occur in the program table.

3.4.3.2 Delete (F2)

This function is to delete useless programs from the system, and they can be main program or sub-program so long as they appear in the table. Steps:

(1) Pressing "PRGRM (program)" to access program management.

(2) Pressing "F2" and the function notes 'delete', then inputting program name which is required deleting.

(3) This program is deleted after pressing "ENTER" key.

Note: Once the program is deleted, it can't be recovered, so it's necessary for you to carefully operate when deleting.

3.4.3.3 Program status

Every program of 21DM system can have four attributes, which are:

A) common b) read only c) hiding program d) read only hiding.

Towards to common attributes, the general edit, delete etc. can be implemented. Towards to read only attribute, the system can be only displayed in the screen for viewing, which the character can't be increased and decreased or deleted wholly. This function can prevent misoperation to corrupt program.

The hiding program can be edited, modified as the common attribute, however the program name isn't displayed in the table, accordingly no one but the operator can operate this program.

The read only hiding is the combination of two attributes of hiding program and read only.

Operating steps:.

(1) Pressing "PRGRM (program)" to access program management.

(2) Pressing " ◄► "

(3) Pressing "F5", and the display function notes 'program status', simultaneously inputting program name in the cursor position.

(4) The "arrow" appears post-program name after pressing "ENTER", which signifies that the program will be assigned attributes.

(5) One of four numbers 0, 1, 2 and 3 is put in the cursor position and pressing enter key, then this program name will be assigned attribute, simultaneously there are also changes in the program name table, and the meanings of four numbers are:.

0: common attribute 1: read only 2: hiding attribute 3: read only hiding.

The default attribute is 0 (common attribute) after editing new program.

3.4.4 Rename, input and output function

The system inputs and outputs program via RS—232 serial port.

3.4.4.1 Rename (F4)

It's used to modify some one program name which is stored in the system program..

(1) Pressing "PRGRM (program)" to access program management.

(2) Pressing“ ◄► ”

(3) Pressing "F4" to display "Rename", simultaneously the cursor appears after the first line 'program'. Please input the program name which will be renamed (has existed in the system), and one "arrow" will appear after this program name after pressing enter key, which signifies that this program name will be renamed to another one, and cursor flashes after the arrow..

(4) Inputting one new program name which must be the nonexistent one, and the original program name will be changed into new name after pressing enter key, simultaneously there are also corresponding changes in the program name table..

The main role of rename sub-function is to change the main program into the sub-program, and the system can only process the main program, the sub-program can only be called by other programs. In order to call one sub-program, it's necessary to implement editing and debugging. The program is changed into sub-program by using "Rename" after there is no error. It can be called by other main programs with necessary sub-program definition statement.

3.4.4.2 Input (F2)

One program can be input from PG machine or other numerical control systems through serial port. . Operation:.

(1) Pressing "PRGRM (program)" to access program management.

(2) Pressing“ ◄► ”

(3) Pressing "F2" to display 'Input', and cursor appears and flashes after the first line 'Program', inputting program name in the cursor position (it must be the program which isn't in the system), and the input process starts after pressing enter key..

(4) The input characters can be seen in turn which are displayed in the screen when communication transfers.

3.4.4.3 Output（F3）

It outputs some one program to the PG machine or other numerical control systems which is stored in the system. . Steps:.

(1) Pressing "PRGRM (program)" to access program management.

(2) Pressing“◄►”

(3) Pressing "F3" to display 'Output', and cursor appears and flashes after the first line 'Program', inputting program name in the cursor position (it has been stored in the system), and the output process starts after pressing enter key..

(4) If the character is typed wrongly when inputting program name, the "F1" key can be used for modification..

(5) Towards to the specific operations which implements program exchanges with computer, please see description files in the communication floppy disk.

3.4.5 List

The list function can rapidly browse and edit user program content in the CNC storage area, even if the hiding program can be displayed under this function. . However, if the

program is read only, then edit can't be implemented, and the operating sequence is as follows:.

(1) Pressing "PRGRM (program)" to access "Program management" mode.

(2) Pressing "F4" to access list sub-function, and displaying the first program in the CNC user program area. This program name (PXX or NXX) is displayed after 'Program' in the first line of the screen, and the program contents are displayed in other regions of screen. . Now, the function keys of F1~F5 are redefined..

F1: Edit. . Pressing "F1"can edit current displayed program contents..

F2: Previous page. . If current program contents are relatively more, which can't be displayed in one screen, and the "F2" key can be pressed to skip to the previous page..

F3: Next page. . If current program contents are relatively more, which can't be displayed in one screen, and the "F3" key can be pressed to skip to the next page..

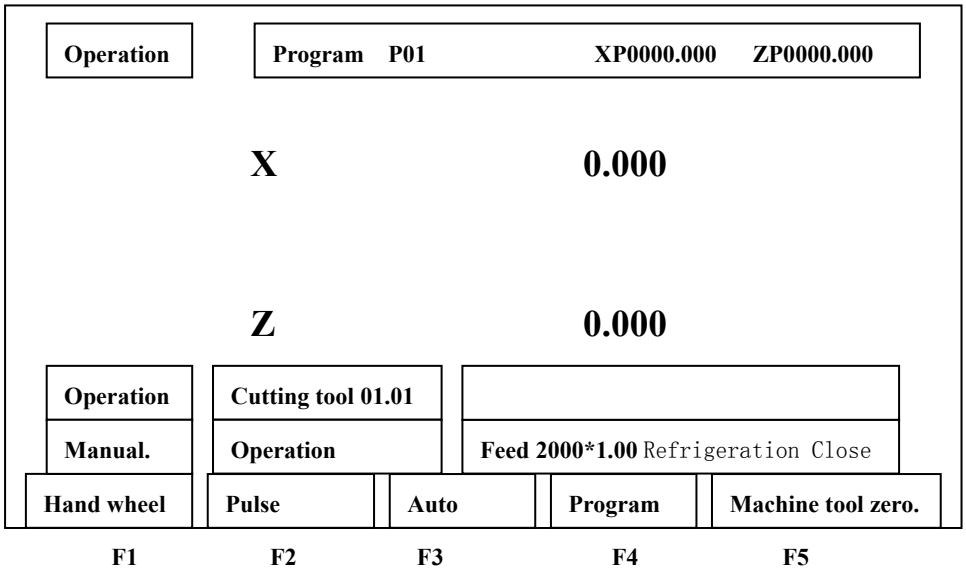
F4: Next segment of program. The next program name and contents of CNC user program area can be displayed by p. ppressing "F4". . If current program is the last one of the user program area, then the first program is displayed after pressing this key..

F5: Previous segment of program. The previous program name and contents of CNC user program area can be displayed by p ressing "F5". . If current program is the first one of the user program area, then the last program contents and name are displayed after pressing "F5".

3.5 OPERT (Process) primary function

The OPERT primary function offers various operations and controls to the system or machine tool, such as auto-cycle, manual continuous feed, selection of feed parameter, MDI mode and so on. Various sub-functions under this primary function are still selected by function keys from F1 to F5..

Pressing "OPERT (Process)" key to access processing primary function, and the displayed image is as follows:



The XP and ZP of first line are the coordinate values which are relative to machine tool zero.. The coordinate value which is displayed with big characters is the one that is relative to programming zero; various functions notes of F function key are displayed in the bottom line; and various states displayed with light background respectively indicate the operation mode,

machine tool status, some parameter values of manual operation such as manual speed, spindle rotating speed and so on.

3.5.1 Auto-cycle (including start processing in the position of any block number)

Auto-cycle is corresponding to automatic mode of machine tool. 'Auto' will be displayed in the operation mode window after pressing "F3" key, then pressing "Save Open program" key. The cursor appears and flashes after 'Program' of the first line in the screen and user can input program name which is ready for operation in this position. "F1" is used to modify wrong-typed characters when inputting program name; the 'program name table' in the middle of screen display existent program name, program size and program attributes in the user program area..

The program name input is right and pressing "ENTER", which describes that system has prepared to execute the program such as pressing "Cycle start", and the program starts operation. When pressing "F2 (line number)", the cursor appears after the program name. It notes user to input line number which is the line to execute line skip. The program before line number is not executed..

This system provides that the programming zero under auto-cycle is the point which the value of system big coordinate is zero, and all programming tracks regard this coordinate as the benchmark. The coordinate zero can be also set with G92 under MDI mode, and the current system coordinate position can also be set as coincidence between processing zero and program zero. . If the big character coordinate displays zero when auto-cycle starts, then the programming tracks which are relative to current cutting tool position . are implemented.

The big character coordinate is not zero when auto-cycle starts, and the system regards that various coordinate axes have moved some distance from zero. The movement amount is the value of the big character coordinate, while the zero of programming coordinate is not the cutting tool position when cycle starts, and zero is the point before movement..

For instance: The programming is:.

G90 G01 X10 Z30 F100

However, when cycle starts, the big coordinate of system is:.

X —50.000

Z 3.500

Then the results of the execution above are: X coordinate moves 30 mm along the positive direction, and Y coordinate moves 26.5 mm along the negative direction, in addition, Z coordinate moves 26.5 mm along positive direction..

After pressing "Cycle start" key, the system firstly implements necessary internal processing such as examination, error detection etc. If error occurs, the note of error will appear; otherwise the program will be executed in order..

Note:(1) The program name of auto-cycle processing will be called out by file open key.

(2) The figure simulation or track will be accessed under automatic processing, and see 3.6 section for specific contents.

3.5.2 Manual operation of machine tool

Manual operation includes two modes of manual continuous feed and stepping feed, which are called manual mode and pulse control mode, and both refrigeration and spindle can be

manually operated..

(1) Manual mode: Manual mode can be accessed by pressing "PRGRM (Program)" key, and under the manual mode, ◄Z-、Z+►、X-▲、X+▼ signify operating keys which are used for various coordinate axes to move along their positive directions or negative directions. When pressing one of them, the corresponding coordinate axis will move long its direction. . The feed speed can be manually set by pressing "FSET". . When pressing one of feed keys above and " **" simultaneously, the speed operation can be set by pressing parameter..**

(2) Pulse control mode: It's same as the manual mode that presses "F2" key to access pulse control operating mode. The coordinate moves some given length along its corresponding direction by every coordinate feed key pressing. This length is set by "ISET"..

(3) Settings for manual operation parameter: They are only valid in manual and pulse control modes..

Pressing "FSET" key: Settings for coordinate movement speed in manual or pulse control modes (letter key region "FSET" key), and the cursor will appear after word 'Feed' with pressing "FSET", then the digits can be input, which signify millimeters for feed per minute. It'll be valid after pressing "ENTER"..

The scope of this speed is 1～6000mm／min（when it's 0.001mm pulse equivalent), if input has error, the system will automatically set for 50.00mm ／min.. "F1" key can be used to modify the wrongly-typed digits..

Pressing "I set" key: Setting up pulse control stepping amount ("I set" key in the letter key region), it's only valid in the pulse control which is used to set up stepping amount.

Pressing "I set" and inputting stepping amount after cursor (0.001～65.5mm)..

Pressing "S set" key: Setting up speed of spindle ("Sset" key in letter key region), and inputting digits in the cursor position which signifies speed of spindle. It'll valid after pressing "ENTER". . The system outputs the analogue which is correspondent to rotating speed. . The top limit of this rotating speed is set with 3# parameter and 4# parameter..

(4) Spindle and refrigeration: In the modes of manual, hand wheel and pulse control, both spindle and refrigeration can be manually operated.

3.5.3 Return to machine tool zero

Under OPERT (Process) function, pressing "F5", the operating mode will become returning to machine tool machine tool zero mode, and three words of 'machine tool zero' will appear after 'Operation'. . "Cycle start" will be pressed and the selected axis will return to zero in the postive direction after pressing "F4" to type in the zero-returning axes X and Z. After pressing "reference point", the system will return to zero according to the principle of "X" axis first then "Z" axis. In the continual repeated processing, in order to eliminate accumulated error, it is recommended that zero-returning be implemented once at intervals. It is recommended that zero should also be returned once after starting up each time, which makes system have one absolute reference benchmark, simultaneously eliminates position shift of various coordinate axes for shutdown last time.

3.5.4 Hand wheel (hand-operated impulse generator)

Towards to lathe control system, hand wheel can control machine tool's rectilinear movement in directions of X or Z. Hand wheel sets up three speed gears (multiply factors) for

various requirement which are respectively X1, X10 and X100. Switch can be implements among speed gears, and the minimum control precision is identical with system control one, while the maximal control speed is 100 times than the pulse equivalent. . Hand wheel is principally designed for fast rectilinear movement, presetting cutter etc. of machine tool..

Hand wheel operation is as follows:.

(1) Pressing "OPERT" primary function key in the main menu to access machine tool operating interface..

(2) Pressing "F1" to access hand wheel operating mode..

(3) Pressing "F4" key to select movement coordinate axis, simultaneously the selected coordinate axes are displayed under 'Operation' prompt in the screen..

(4) Pressing " **" key to select hand wheel multiply factor, simultaneously the selected multiply factor is displayed under 'Operation' prompt in the screen.**

(5) Turning hand wheel, and the machine tool makes response movement.

(6) To quit hand wheel status, either "F" function key (F1—F5) can be pressed.

3.5.5 System status setup

Under 'Auto' sub-function of "OPERT (Process)", there are two statuses setting up related with machine tool-numerical control system..

(1) Single segment: Pressing "F1" key for toggle. When it's valid, 'single segment' is displayed after 'Operation'. The system will stop to wait for user to input with executing block of each time. Pressing "Cycle start" key for each, the system will execute one block downwards. If the "Cycle cancel" key is pressed when waiting, this cycle will be cancelled and can't be recovered..

The single segment points to every movement related with machine tool control, not mere one line of the program. For instance: If spindle setup and coordinate movement are included in certain line program, this line program includes two executable movements. The last program completes and waits, pressing "Cycle start" once; the spindle operates and waits again. If "Cycle start" is pressed again, the coordinate will move.

(2) Pause: It's correspondent to "Cycle start" that auto-cycle can be paused and processed by "Cycle pause". . Pressing "Cycle start" after pause to continue, if pressing "Cycle cancel", this automatic processing will be cancelled..

Note: The track display function can be accessed after automatic processing starts.

3.5.6 MDI operating mode

In the manual, auto-pulse control and hand wheel modes of "OPERT (Process)" primary function, the cursor appears in the second line of the screen with pressing "GMDI", now one line program can be typed in and the system will be executed after pressing "ENTER" key. This segment of program requires no block number.. In the automatic mode, all executable blocks can be executed in MDI mode.

3.6 Figure display function

Under "OPERT (Process)" primary function, the automatic mode can be accessed by pressing "F3 (Auto)", and now pressing "F5 (track display)" to access figure display mode (online or simulation).. This function is used to display movement track of tool nose under certain processing program control. This function can make user directly view the operational

process of programming track, and at the same time, big errors in the program can be evidently found combining coordinate values which are displayed in the screen. The figure in the screen can track tool nose's cutting movement in synchronization, and displays appearance of bar. The operator can directly view the manufacture course of revolving body from the screen when tool nose moves. In the simulated condition, the cutting tool center track is displayed in the screen, however various coordinate axes of machine tool don't move, in addition, control functions of various machine tool electrical equipments are invalid which are mainly for debugging program. When program has no errors, it can be processing to avoid failures even accidents owing to programming oversight..

3.6.1 Image access sequence of figure display function

(1) Pressing "OPERT（Process）" key

(2) Pressing "F3（Auto）" key

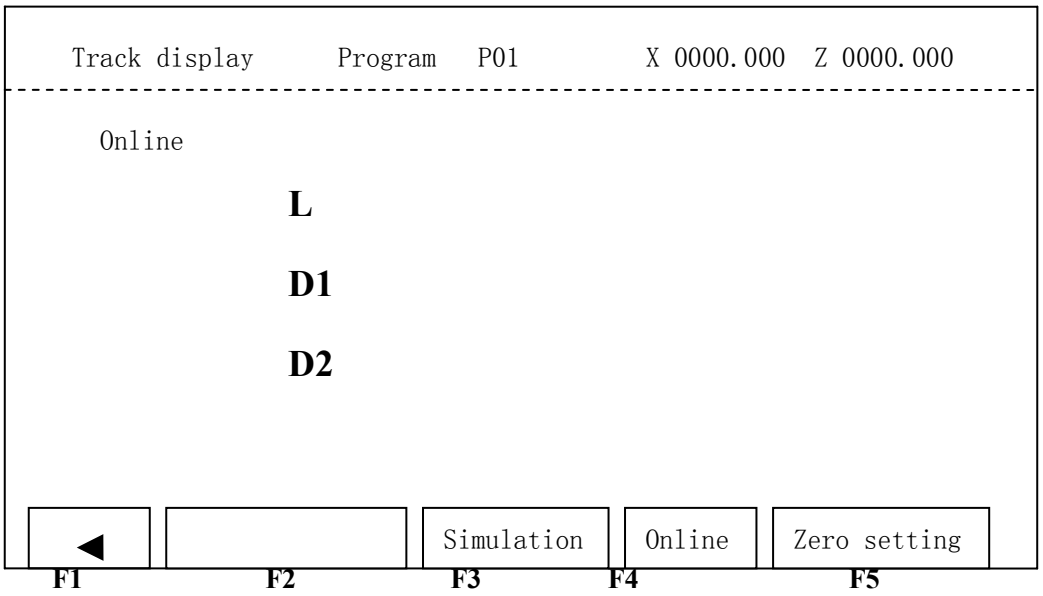
(3) Pressing "Save Open program" key

(4) Inputting file name "PXX"

(5) Pressing "ENTER"

(6) Pressing "F5（track display）"

Screen displays as follows:.



3.6.2 Image selection of figure display function

After accessing figure display function image according to 3.5.1 sequence, the function keys of F1～F5 will be defined..

F1: It's used to delete wrong digit input..

F3: (Simulation) There is only figure display, and cutting tool won't move..

F4: (Online) The figure display and cutting tool movement are in synchronization..

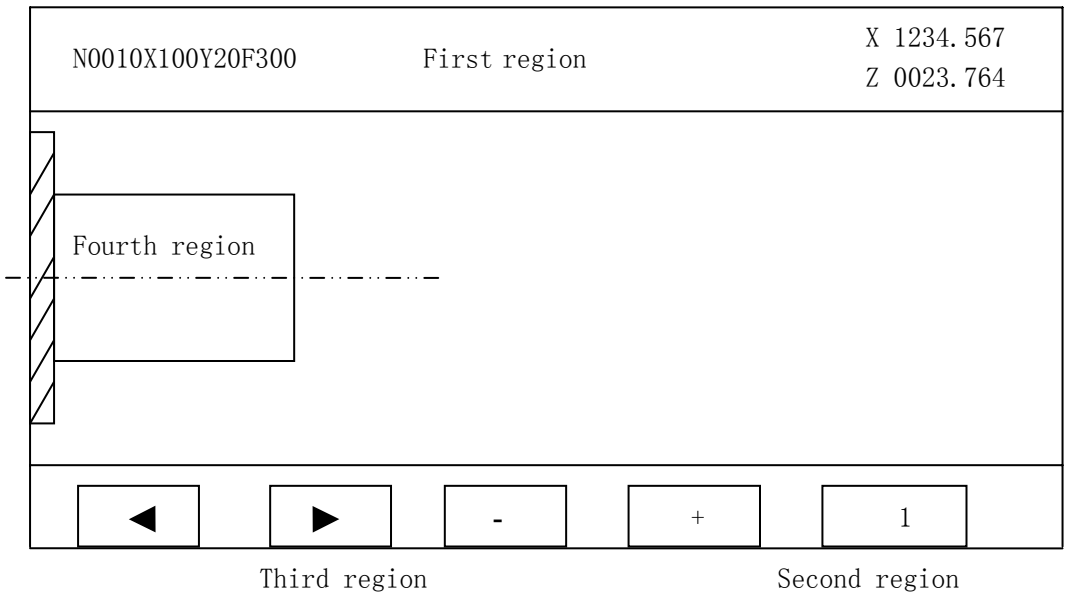
F5: (Zero setting) It's used to set work piece zero, which makes it coincide with programming zero..

L: Inputting rough length.

D1: Inputting rough external diameter

D2: Inputting rough internal diameter.

First the cursor flashes after L, and user can input L value, which the scope is 0.001-9999.999 mm. Its input error can be modified with "F1" to move cursor, and it's valid by pressing "Enter" after input completes. At the same time, the cursor moves to D1 and inputs rough external diameter value, and it's valid after pressing "ENTER", inputting the rough internal diameter D2. If it's right after pressing "ENTER", the screen will display another image as follows, which is the frame of image. The system always sets the whole rough within scope of 300X200 (the whole screen is 320X240). . Accordingly, towards to the same program, the bigger L and D are, the smaller the figure is..



First region: It's used to display current processing program segment..

Second region: It's used to display pixel number for each movement of supposed cutting tool .

Third region: F key function note.

Fourth region: It's for screen figure display, and the processing track can only be displayed in this region, the exceeding segment can't be displayed..

There is one vertical line in the right lower part of figure display region, and upper end of it signifies the cutting tool center. Because the system doesn't learn about movement direction and course of track, so user must move small cross line to select one proper position in the screen where it stops, which is the work piece coordinate point currently displayed..

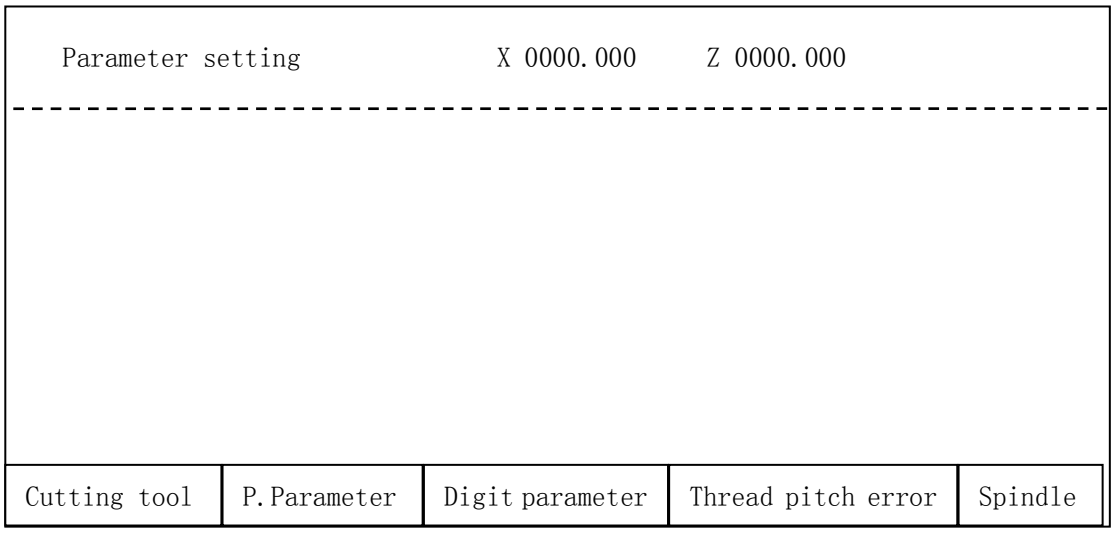
F1, F2, F3 and F4 can be used to move vertical, and the movement direction is the arrow one which is correspondent to F1—F4 in the screen. While pressing F function key every time, the movement amount of vertical can be one pixel/five pixels. . How to select movement amount is determined by F5. Two digits of 1 and 5 alternately appear in the second region of the screen with pressing "F5". Therefore F5 can move the vertical center to the optional position of figure display area through F1-F4..

Once vertical is in place, the program will start execution with pressing "Cycle start" key. . The track starts from the upper end of vertical, and the programming track is simulated according to program requirements. . However, whether the machine tool operates d. epends on drive power and simulation selection or online. . When program execution completes, other primary function keys can be pressed to quit figure track display. . After simulation completes, whether the program is right can be judged through figure. . If there is error, pressing "PRGRM (Program)" primary function key to return to edit mode for program modification. When the program modification completes, the figure simulation will be implemented again until it's right. After simulation completes, the figure track display can only be quitted

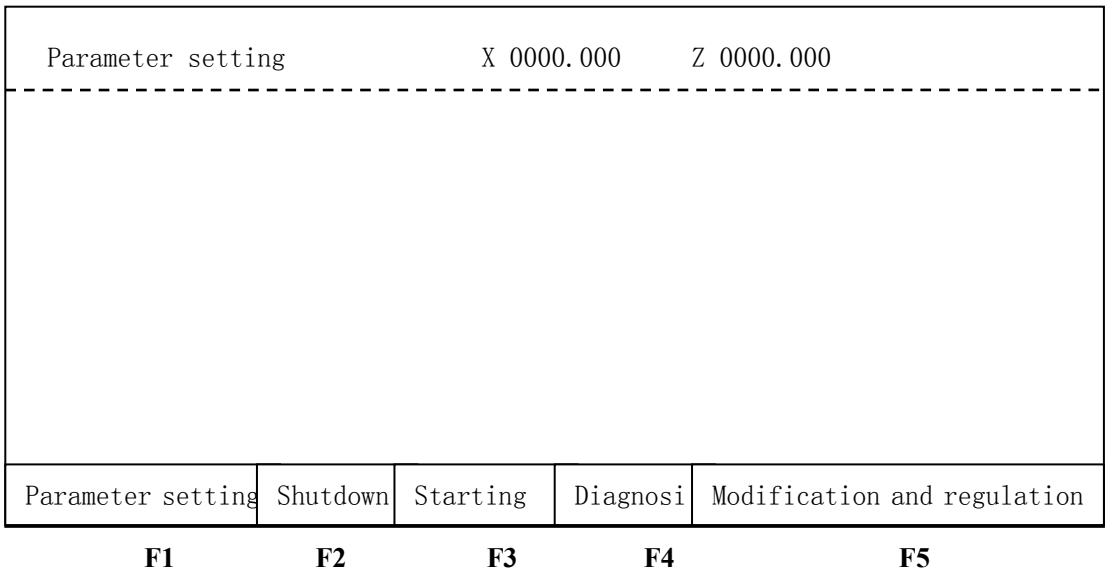
Chapter Four Parameter function

4.1 31DT parameter system

The PARAM (Parameter) primary functions are the parameter setting and status display of machine tool-numerical control system. As one of numerical control systems, there are some parameters which should be set up by users. Under the PARAM (Parameter) primary function, the available sub-functions can also be selected by F1～F5 keys, and the PARAM (Parameter) image accessed is as follows:.



(Pressing "◄►" key to select another image)



System zero setting.

31DM parameters can be divided into three categories below:

(1) Examination category: Testing whether the external input port status, spindle rotating

speed and spindle encoder operate normally..

(2) System setup category: Initializing system, setting up password and formating electronic disk and so on..

(3) Parameter system:.

① System parameter: (P parameter) 00#—89#.

② Cutting tool compensation parameter: 10 cutting tool numbers.

③ Thread pitch error compensation: 150 points/axis, three axes of X, Y and Z in total..

④ Digit parameter: 8-bit parameters of 30 and 240 statuses.

⑤ Starting up/shutdown: recover/save system status (optional)

4.2 Basic conception of parameter

Before introducing parameter, some basic conceptions should be introduced first.

4.2.1 Time constant of speed increase and decrease

It's the time that the operating speed consumes from zero to maximum set one, or from maximum set speed to stop towards to the coordinate axis of machine tool.

4.2.2 Acceleration

The 31DT system adopts sampling control method and implements sampling control for one time to various coordinate axes. The speed increase and decrease of various axes adopt control method of even acceleration in every other 4.096 millisecond, namely the change of motor speed (if it operates) of each sampling interval (4.096ms) is constant value, which is the speed increase and decrease of the system. Its calculating formula is:.

$$a = \frac{V_m \times Ts^2}{60 \times T_m}$$

Among it: Ts: System sampling time (4.096 ms).

Tm: Time constant of speed increase and decrease (ms).

Vm: Maximal speed (mm/min).

A: Acceleration.

The acceleration can be changed through changing maximum speed or changing speed increase and decrease..

Generally, the maximum speed is used to limit the one in this movement status, and the acceleration value can be determined with time constant of speed increase and decrease, while the actual operating speed of system should be less than or equal to the maximum speed.

V_m	↑	↓	–	–
T_m	-	-	↓	↑
a	↑	↓	↑	↓

The relationship among acceleration a, maximum speed Vm and time constant Tm:. "—" signifies constant, while h and Vm change at the same time, it's specifically determined by the formula.

4.2.3 Electronic gear ratio

Regulating system output which makes the coordinate operating value of system concord with the distance of table movement, and it's not necessary to regulate transmitting ratio of motor and screw for this..

The electronic gear is set by system parameters. Two parameters of each axis can be called multiply factor MLT and percentage DVT, and when the bearing ball screw transmitting working table operates.

$$\frac{MLT}{DVT} = \frac{Pmt \times Gf}{Pcn \times Gd}$$

Among it: Pmt: Pulse number of each rotation for motor, and the stepping motor: Pulse of each rotation.

Servo motor: Encoder line number X encoder multiple frequency number.

Pcn: Screw thread pitch μm

Gf: Total of driven wheel teeth number in the transmitting between motor and screw connection.

Gd: Total of driving wheel teeth number in the transmitting between motor and screw connection.

Towards to direct connection，Gf＝Gd＝1

MLT and DVT must take the smallest positive integral value.

The scope of MLT and DVT is 0—65535, but the ratio must within 0.01—100.

4.2.4 Parameter password

For preventing parameter modification without intention, the system sets up the password. First the password unlocking must be input, and then the parameter modification can be made, which the method is as follows:.

(1) Pressing "Cutting tool modification and regulation", and the cursor appears after "Password".

(2) Inputting password which concords with the password in the "System zero setting" function.

(3) Pressing "ENTER". If input is right, the system will clear away this line else display error.

4.3 System parameter (P parameter)

In order to modify the system parameter, the right password according to 4.2.4 must be input first, then:.

(1) Pressing "PARAM (Parameter)" to access parameter image;.

(2) Pressing "F2 (P parameter)" to access P parameter (system parameter) and input password;.

(3) Pressing "F1～F4" to move cursor to corresponding parameters;.

(4) Pressing numerical key can directly implement input again;.

(5) Pressing enter key "ENTER" to confirm the input;.

(6) Pressing "Save Open program" to save parameters into the electronic disk;.

Cutting tool zero function:.

One position can be selected optionally on the machine tool as the processing start point (cutting tool zero), and this point has two groups of coordinates:.

(1) The big coordinates based on work piece coordinate system..

(2) The absolute coordinates based on machine tool reference point.

F5 can store the two groups of coordinates above in the current position of tool nose..

Note: The absolute coordinates can be meaningful only after machine tool returns to reference point; else the operation will be disordered..

When machine tool coordinate moves to optional position, both G75 and G76 instructions can be used to return to the set processing start point..

(1) Based on the machine tool coordinate , G74 is generally used to return to the reference point , then returning to processing start point with G75..

(2) Based on the big coordinate of working coordinate system, G76 can be used to return to the processing start point..

G75 instruction can eliminate accumulated errors in the work process; however machine tool must have machine tool reference point..

G76 requires no machine tool reference point; however it can't eliminate accumulated errors..

The P parameters have 90 in total from 00# to 89#. 30 can be displayed in every screen which can be switched with "ALT" key.

4.3.1 Meaning of P parameter

Parameter number	Scope	Factory set
00	G00 Fast locating speed setting (mm/min)	6000
01	Holder reverse lock time set (second)	0.8
02	The cutting tool number of machine tool	4
03	The spindle 1 gear rotating speed top limit (rpm) (used for frequency converter analogue output)	3000
04	The spindle 2 gear rotating speed top limit (rpm) (used for frequency converter analogue output)	1000
05	＝0 Machine tool of electro-holder; ＝1machine tool array and change	0
06	X axis direction clearance (1 min)	0
07	Z axis direction clearance (mm)	0
08	The coordinate of X-axis presetting cutter point which is relative to machine tool zero	0
09	The coordinate of Z-axis presetting cutter point which is relative to machine tool zero	0
10	Manual fast speed (mm/min)	5000
11	Spindle reversing time delay (second)	1
12	Spindle braking time delay (second)	1
13	Spindle starting time delay (second)	1
14	M05 relay pulse output time delay (second)	1
15	Relay pulse output time delay (M71—M85)(second)	1
16	Screw in and screw out speed in the thread X direction (see G86 description for detailed information)	5000
17	Initial image, ＝0 version;＝1 manufacturer description (option); ＝2 operating image	0

Parameter number	Scope	Factory set
18	X-axis presetting cutter point set value which is relative to work piece coordinate	0
19	X-axis presetting cutter point set value which is relative to work piece coordinate	0
20	Spindle pulse number per rotation	1200
21	M04 output =0: maintaining output; ＝1: pulse output	0
22	＝0：M05 close S1、S2、S3； =1：M05 doesn't close S1、S2、S3	0
23	Percentage of unstable spindle speed, the thread can be processed when it's less than this value	5
24	Undetermined	0
25	The smoothing tool amount of the last cutting when processing thread,=0: without smoothing tool (mm)0	0
26	Undetermined	
27	The segment number increments 0—99 are automatically generated when editing program; the segment number is not generated when it's equal to 0	10
28	The low limit of spindle rotating speed in the time of constant linear speed cutting.	100
29	The brightness value of LCD when starting up (10—32)	14
30	The time constant of X direction when it's G00 (millisecond)	400
31	The time constant of Y direction when it's G00 (millisecond)	400
32	The time constant of Z direction when it's G00 (millisecond)	400
33	The electronic gear multiply factor of X direction	1
34	The electronic gear percentage of X direction	1
35	The electronic gear multiply factor of Y direction	1
36	The electronic gear percentage of Y direction	1
37	The electronic gear multiply factor of Z direction	1
38	The electronic gear percentage of Z direction	1
39	The time constant of various axes when cutting operates (millisecond)	400
40	The time constant of Z axis when processing thread (millisecond)	400
41	Speed top limit when Cutting	5000
42	Speed low limit when G00 is fast	0
43	Looking for slow speed of zero signal when returning reference point	50
44	The time constant of each axis of soft limit to stop	200
45	Speed top limit for thread cutting (Z direction)	3000
46	Input signal de-jitter number	3
47	Time delay between electro-holder positive and reverse rotation	0.1
48	Speed top limit of clearance compensation (time constant is same as 39#)	2000
49	The time constant of X direction screw in and screw out de-trailing when processing thread	400
50	Reverse clearance value of Y axis	0

Parameter number	Scope	Factory set
51	The time constant for various axes of hand wheel operating	400
52	The interval length for thread pitch error compensation of X-axis	0
53	The point number of X-axis thread pitch error compensation	0
54	The interval length for thread pitch error compensation of Y-axis	0
55	The point number of Y-axis thread pitch error compensation	0
56	The interval length for thread pitch error compensation of Z-axis	0
57	The point number of Z-axis thread pitch error compensation	0
58	The limit for contour error of circular interpolation (mln)(0.002 in general)	0.002
59	The screw in/de-trailing speed top limit of X direction when processing thread	3000
60	The positive direction soft limit coordinate of X-axis from the reference point	0
61	The negative direction soft limit coordinate of X-axis from the reference point	0
62	The positive direction soft limit coordinate of Y-axis from the reference point	0
63	The negative direction soft limit coordinate of Y-axis from the reference point	0
64	The positive direction soft limit coordinate of Z-axis from the reference point	0
65	The negative direction soft limit coordinate of Z-axis from the reference point	0
66	The input terminal number of X-axis reference point coarse positioning signal	12
67	The input terminal number of Y-axis reference point coarse positioning signal	12
68	The input terminal number of Z-axis reference point coarse positioning signal	12
69	The input terminal number of X-axis reference point fine positioning signal	9
70	The input terminal number of Y-axis reference point fine positioning signal	15
71	The input terminal number of Z-axis reference point fine positioning signal	18
72	Positive direction limit input terminal number	17
73	Negative direction limit input terminal number	21
74	External general alarm input terminal number	8
75	Drive alarm input terminal number	16
76	Spindle high/low speed input terminal number	22
77	The spindle 3 gear rotating speed top limit (rpm) (used for frequency converter analogue output)	1000
78	The spindle 4 gear rotating speed top limit (rpm) (used for frequency converter analogue output)	1000

Parameter number	Scope	Factory set
79	The top limit for holder positive rotation (44# alarm occurs at the start of holder positive rotation)	4 seconds
80	Position ring constant m (0—50)	0
81	Position ring constant KP (5—100)	80
82	Position ring constant KD (0—50)	0
83	The spindle reversion time delay at the time of rigidity threading	0.5 millisecond
84	Self-specified thread tooth type angle I, used for single side thread cutting.	0.5 millimeter
85	Self-specified thread tooth type angle II, used for single side thread cutting.	0
86	Speed for each axis of returning to machine tool zero	3000
87	The feed length increased of each millimeter at the time of rigidity threading	0
88	The feed speed of thread X direction	2000
89	The waiting time of feed axis angle-specified stop (millisecond)	20.000

4.4 Digit Parameter

The digit parameter can be used for setting towards the one which has only two selective conditions and branches. Each parameter has 8 digits and each digit has only two states of 0 or 1, which can be used for switch selection of certain status, and the system has 30 digit parameters in total which determines 240 states.

4.4.1 Access

Key-press sequence "PARAM （Parameter）"→"◄►"→ " F3 （digit parameter）".

4.4.2 Digit parameter setting

00#

D/R	L/M	SLOF	L3TH		ZGR	YGR	XGR

D／R：=1：Radius programming =0：Diameter programming.

L／M：=1：The function of milling machine interface =0：The function of lathe interface.

SLOF：=1： When the program execution completes, M05 and M09 are not inserted automatically.

=0： When program execution completes, M05 and M09 are inserted automatically..

L3TH：=1：the third movement axis are opened under lathe execution mode，whose function is valid when L／M=0..

ZGR：=1：Z axis opens electronic gear function

= 0 ： Z axis doesn't open electronic gear function..

YGR：=1：Y axis opens electronic gear function

= 0 ： Y axis doesn't open electronic gear function..

XGR：=1：X axis opens electronic gear function

= 0 ： X axis doesn't open electronic gear function..

00# factory set is 00000000.

01#.

REDP					**ZPCE**	**YPCE**	**XPCE**

REDP：=1：Dynamic thread pitch error compensation values are displayed on the operation interface，and their values are respectively signified with XH、YH and ZH.

ZPCE：=1：Z axis opens thread pitch error compensation function..

YPCE：=1：Y axis opens thread pitch error compensation function..

XPCE：=1：X axis opens thread pitch error compensation function.

01# factory set is 00000000.

02#.

BKDP	**SLE**	**SLE**	**SNZ**		**ZBKE**	**YBKE**	**XBKE**

BKDP：=1：The reversion clearance compensation condition is displayed dynamically on the operating interface, and signified with XK 、YK 、ZK.

SLE： =1：Open soft limit function.

SLS： =1：Each axis stops suddenly at the soft limit.

=0：Each axis stops with speed reduction（recommendation）.

SNZ： =1：It's not necessary for soft limit to be valid returning to reference point

=0：It's valid that the soft limit must return to the reference point..

ZBKE：=1：Z axis opens reversion clearance compensation.

YBKE：=1：Y axis opens reversion clearance compensation.

XBKE：=1：X axis opens reversion clearance compensation.

02# factory setting is 00000111.

03#

RDYE	**RDYK**	**FTFE**			**ZREF**	**YREF**	**XREF**

RDYE：=1：Self-verifying is passed after starting up, and CNC outputs ENABLE signal to the servo driver （each axis and contact point output）.

RDYE：=0：self-verifying is passed after starting up, and CNC doesn't output ENABLE sign to the servo driver （each axis and contact point output）.

RDYK：=1：The "RDY" key on the operating keyboard can output the ENABLE signal or not.

= 0： The RDY key is invalid.

FTFE： =:1 When G00 fast moves, the speed modification and regulation is valid but it cannot exceed the top limit of fast speed which is determined by the 00# Parameter..

=0： G00 zero speed modification and regulation..

ZREF： =1：The function of Z axis returning to the reference point is opened,

=0：The function of Z axis returning to the reference point is unopened..

ZREF ： =1：The function of Y axis returning to the reference point is opened,

=0： The function of Y axis returning to the reference point is unopened..

XREF： =1： The function of X axis returning to the reference point is opened,

=0：The function of X axis returning to the reference point is unopened..

03# factory set is 10000111.

04#

HL08	**HL07**	**HL06**	**HL05**	**HL04**	**HL03**	**HL02**	**HL01**

HL0x（x=1-8）04# factory set is 00000000

05#

HL16	**HL15**	**HL14**	**HL13**	**HL12**	**HL11**	**HL10**	**HL09**

HLxx （xx =09-16）respectively signifies whether it's high level (=1) validity or low level (=0) validity for the input level of the input terminal which is same as the number after HL..

05# factory set is 00000000.

06#

HL24	**HL23**	**HL22**	**HL21**	**HL20**	**HL19**	**HL18**	**HL17**

HLxx （xx =17-24）respectively signifies whether it's high level (=1) validity or low level (=0) validity for the input level of the input terminal which is same as the number after HL..

06# factory set is 00000000.

07#

HL32	**HL31**	**HL30**	**HL29**	**HL28**	**HL27**	**HL26**	**HL25**

HLxx （xx =25-32）respectively signifies whether it's high level (=1) validity or low level (=0) validity for the input level of the input terminal which is same as the number after HL..

The WA—31DT series have no such input terminal, and requires extension.

07# factory set is 00000000.

08#.

HLS4					**XINV**	**YINV**	**ZINY**

HLS4：=1 spindle M function is output with 4-gear speed rotation and the 4-gear speed is selected by M41-44, which is suitable for converter with mechanical gear of the 4 gear. The 4-gear speed is determined by the parameters of 3#、4# an d 77#..

=0： Select spindle high and low signal, which is suitable for the spindle converter with high and low speed mechanical conversion, and high and low signal is input into the system. The analogue top limit for 3# or 4# of P parameter is determined by the spindle high and low signal..

XINV：=1：The reversion movement of , =0：The positive movement of YINV ： =1：

ZINV：=1：The reversion movement of 08# factory set is 00000000.

09#

SSN	**SCOR**	**OVS**			**TZR**	**TYR**	**TXR**

SSN： =1： Open position ring regulator , and KD 、KP 、KI are valid. =0： It is not opened..

SCOR：=0： Soft limit is determined by the machine tool coordinates =1：The soft limit is determined by the work piece coordinates..

OVS： =1：When the hard limit switch is pressed, it stops with speed reduction.

=0：The hard limit switch stops suddenly when it is pressed..

TZR: =1: When the CNC is powered on, it will send ENABLE contact signal to servo output, later checking whether the READY signal of Z direction servo is received. If it's not received, the error of "Drive is not ready" is displayed on the operating interface..

=0: The READY signal of servo isn't checked..

TYR: =1: When the CNC is powered on, it will send ENABLE contact signal to servo output, later checking whether the READY signal of Y direction servo is received. If it's not

received, the error of "Drive is not ready" is displayed on the operating interface..

=0: The READY signal of servo isn't checked..

TXR: =1: When the CNC is powered on, it will send ENABLE contact signal to servo output, later checking whether the READY signal of X direction servo is received. If it's not received, the error of "Drive is not ready" is displayed on the operating interface..

=0: The READY signal of servo isn't checked..

The factory set of 09# is 00000000

10#

FSMT	**SEN**	**DNCT**	**DNCS**	**SIGN**	**ZROT**	**YROT**	**XROT**

FSMT: =1: Low vibration mode when the feed is selected.

=0: Selecting high precision mode of speed.

SEN: =1: The S function is disallowed to implement S1, S2, S3 and S4 to control the spindle 4 gear electrical shift.

=0: The S function is allowed to implement S1, S2, S3 and S4 to control the spindle 4 gear electrical shift..

DNCT:=1: The data reception speed is Ⅱ gear when DNC processes..

=0: The data reception speed is I gear when DNC processes.. （Recommendation）

DNCS:=1: When DNC starts, it first sends XON character to the upper machine tool.

=0: DNC doesn't send XON character..

SIGN:=1: The first line of numerical control program has no % when DNC processes..

=0: The numerical control begins with % when DNC processes..

XROT:=1: The X direction is signified with circumference, 0—360 degree

=0: The X direction is signified with length.

YROT:=1: The Y direction is signified with circumference, 0—360 degree

=0: The Y direction is signified with length.

ZROT:=1: The Z direction is signified with circumference, 0—360 degree

=0: The Z direction is signified with length.

The factory set of 10# is 000000.

11#

JET	**ALRM**	**DANS**			**ZSRF**	**YSRF**	**XSRF**

JET: =1: Special machine tool control (the function is determined according to the special machine quality)..

=0: General system control..

ALRM:=1: Emergency stop/M77 is output at limit, (the alarm lamp can be controlled).

=0: Don't output alarm signal..

DAVS:=1: The spindle analogue selects 0—5V.. =0: The spindle analogue selects 0—10V.

XSRF: =1: One switch that X direction returns to reference point, it's the coarse positioning when pressing and fine positioning when releasing.

=0: Two switches that X direction returns to reference point, and the coarse positioning and fine positioning are separated.

YSRF:=1: One switch that Y direction returns to reference point, it's the coarse positioning when pressing and fine positioning when releasing.

=0: Two switches that Y direction returns to reference point, and the coarse positioning and fine positioning are separated.

ZSRF:=1: One switch that Z direction returns to reference point, it's the coarse positioning when pressing and fine positioning when releasing.

=0: Two switches that Z direction returns to reference point, and the coarse positioning and fine positioning are separated.

When selecting one switch, the coarse and fine positioning switches of this axis must be same as the input terminal number, namely in the P parameter:.

P66=P69，when XSRF=0

P67=P70，when YSRF=0

P68=P71，when ZSBF=0

The factory set of 11# is 00000000

12#

WHLS		**CH/EN**					**TBCD**

WHLS:=1: Hand-operated pluse generator (multiply factor, axis selection) is selected by the external switch..

=0: Selecting system keyboard to control hand-operated pulse function..

CH／EN: =0: Chinese interface

=1: English interface (the English version software must be replaced).

TBCD:=1:The cutting tool number outputs BCD code.. =0: The normal electro-holder response..

The factory value of 12# is 00000000.

13#

EM03	**ES4**	**S4DF**			**ZFOR**	**YFOR**	**XFOR**

EM03=0: The spindle S programming, it's valid with M03.. =1: The spindle S programming, it's valid without M03..

ES4 =0: 31DT system has no S4 function (only S1, S2 and S3) =1: 31DT system has S4 function..

S4DF=0: Defining M78 as S4 output (it's only valid when ES4=1)

=1: Defining M79 as S4 output.

XFOR: They are zero-returning directions of X, Y and Z axes respectively..

YFOR＝0: Along the positive direction of each axis..

ZFOR＝1: Along the negative direction of each axis..

4.5 Thread pitch error compensation

Each axis of the system can be input most 150 error compensation points, and the compensation point number of each axis and interval between two points are determined by P parameter 52#—57#. The point system exceeding compensation scope regards the thread pitch error as 0, and the system between two compensation points . regards the thread pitch error as linear change..

(1) Access: Pressing "PARAM (Parameter)", then pressing "F4".

(2) Pressing "F1 —F4" to move cursor can select error value of some one point, while the coordinate value of this point which is relative to machine tool coordinate (machine tool reference point) is in the lower part of the screen, and it's signified by XP, YP and ZP..

(3) Pressing "ALT" key can skip to the next page;.

(4) Pressing "F5" can select axes of X, Y and Z.

4.5.1 The required caution problems of thread pitch error compensation

(1) The password must be input and then modification can be implemented..

(2) The input value is point measurement error, namely the compensation value which must be input to offset this error..

(3) Whether thread pitch errors of various axes implement compensate is determined by 01# digit parameters Zpce, Ypce and Xpce..

(4) The dynamic compensation condition can set the REDP digit of 01# digit parameter as 1, and the compensation value in the operating process of various axes can be dynamically displayed on the operating interface..

(5) 21DM system must first return to machine tool zero, then the thread pitch compensation can be implemented..

(6) After returning to machine tool zero, the small coordinate in the top right corner is displayed as: XP 00000.000 YP 00000.00 ZP00000.000 require compensation, and XP, ZP coordinates must move to the negative direction..

(7) The thread pitch error compensation interval length and compensation point number of X direction and Z direction can be determined according to screw length, screw precision and processing work piece size of them. Every axis most compensates 150 points (52, 53, 56 and 57parameters)..

(8) Using laser interferometer to check errors of every other distance (52, 56 parameters) along negative direction from machine tool zero in the three directions X and Z..

(9) Inputting the measured errors into the thread pitch error item in the parameter table..

(10) Setting the digit parameter 03# as: XXXXX101(open reference point returning function, and X signifies that this digit can be I / 0)

4.5.2 Thread pitch error compensation example

Example: The effective length of X screw rod is 300mm, which totally compensates 100 points. 52# parameter＝300 / 100＝3, 53# parameter＝100 (≤150).

Utilizing laser interference to measure the thread pitch error:(returning to zero first which makes XP＝0).

Moving to -3mm position along -X direction (system display), and moving to -2.974 with actual measurement, which 1# thread pitch error is -0.006.

Moving to -6mm position along -X direction (system display), and moving to -6.003 with actual measurement, which 2# thread pitch error is +0.003.

Moving to -9mm position along -X direction (system display), and moving to -9.007 with actual measurement, which 3# thread pitch error is +0.007.

Moving to -12mm position along -X direction (system display), and moving to -11.990 with actual measurement, which 4# thread pitch error is -0.010.

Moving to -15mm position along -X direction (system display), and moving to -14.998 with actual measurement, which 5# thread pitch error is -0.002.

Moving to -18mm position along -X direction (system display), and moving to -17.991 with actual measurement, which 6# thread pitch error is -0.009.

Moving to -21mm position along -X direction (system display), and moving to -21.001 with actual measurement, which 7# thread pitch error is +0.001.

Moving to -24mm position along -X direction (system display), and moving to -24.002 with actual measurement, which 8# thread pitch error is +0.002.

Moving to -27mm position along -X direction (system display), and moving to -27.009 with actual measurement, which 9# thread pitch error is +0.009.

Moving to -291mm position along -X direction (system display), and moving to -291.011 with actual measurement, which 97# thread pitch error is +0.011.

Moving to -294mm position along -X direction (system display), and moving to -294.000 with actual measurement, which 98# thread pitch error is 0.

Moving to -297mm position along -X direction (system display), and moving to -296.999 with actual measurement, which 99# thread pitch error is -0.001.

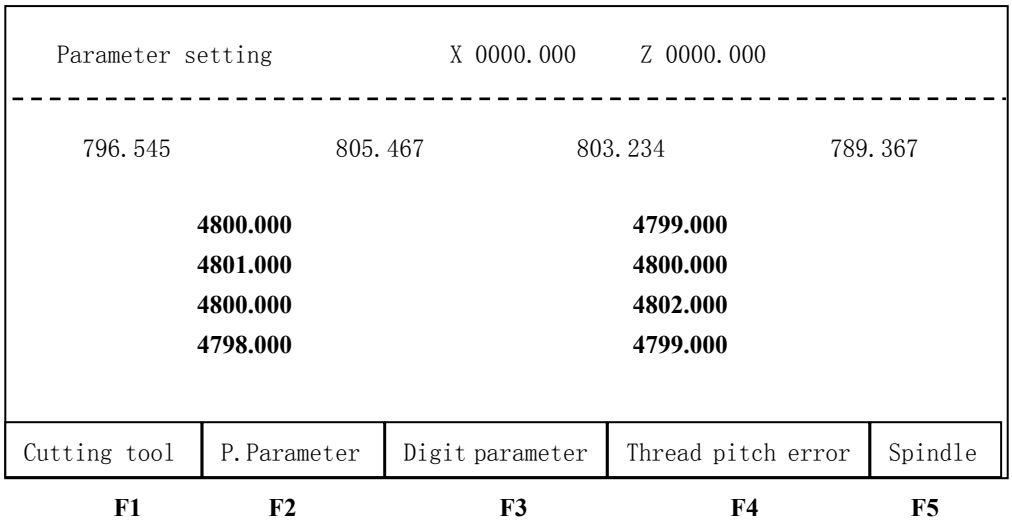
Moving to -300mm position along -X direction (system display), and moving to -300.007 with actual measurement, which 100# thread pitch error is +0.007.

Finally saving the data and CNC system will automatically implement thread pitch compensation when processing.

4.6 Spindle

Pressing "F5" key to access this sub-function. . It inspects whether the spindle encoder is normal as well as measures and displays current spindle rotating speed.. Accessing this function when spindle rotates, CNC displays the checked spindle speed and pulse number X4 sent by encoder in every rotation. Normally the encoder of 1200 lines is 4800, 3 to 5 pulse deviations are allowable, but if it's too more at a time, it must be decreased in the adjacent previous/next time without accumulated error.. It's normal that the displayed data is probably incorrect of the beginning 1 to 3 times.. The checking stops when display is full of screen.. Except for coordinate movement direction key and primary function key, pessing other optional key to continue checking.

As the figure below:.



4.7 Cutting tool parameters

About the effect of cutting tool parameter in the programming, please see 1.2.6..

Cutting tool compensation operation:.

(1) Pressing "F1 key" to access cutting tool parameter setting menu, and at this time the cursor flashes in the parameter position of 'T01' cutting tool number..

(2) At this time, the F function key is redefined, and pressing "F1" or "F2" keys to move

cursor to the required cutting tool nunmber position..

(3) If the X direction value is to be input, pressing "F3 (cutting tool compensation)" then directly inputting work piece diameter of trial cut, finally pressing "ENTER" to generate the cutting tool compensation..

(4) If the Z direction value is to be input, first pressing "ENTER" and pressing "F3 (cutting tool compensation)"; then inputting work piece length of trial cut, finally pressing "ENTER" to generate the cutting tool compensation..

(5) Pressing "Save Open program" key to save the input or modified parameters..

Modification and regulation operation:.

See 2.4.3.9.

Input operation:.

See 2.4.3.8

4.8 Coordinate modification and regulation

After accessing "PARAM (Parameter)" primary function, pressing "◄►" key and "F4 (modification and regulation)" key, then coordinate modification and regulation interface appears.. See 2.4.5 for detailed information.

4.9 Diagnosis (external input signal monitoring)

After accessing "PARAM (Parameter)" primary function, the "◄►" key should be pressed then the "F3 (Diagnosis)" key. This function will display status of external input port, such as whether there is signal input into the system, or whether external limit, zero signal are valid and so on. When certain digit displays 0, no signal is input, while it becomes 1 which signifies there is signal input of corresponding input port. As is the figure below:.

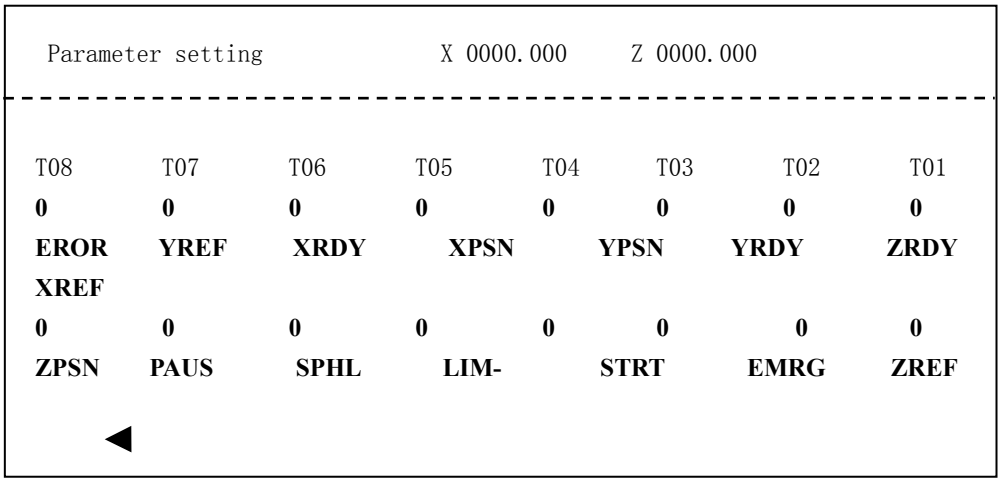

Parameter setting
X 0000.000
Z 0000.000
T08 T07 T06 T05 T04 T03 T02 T01
0 0 0 0 0 0 0 0
EROR YREF XRDY XPSN YPSN YRDY ZRDY
XREF
0 0 0 0 0 0 0 0
ZPSN PAUS SPHL LIM- STRT EMRG ZREF
◄
F1 F2 F3 F4 F5

图 3-6

Among it: T01—T08 are correspondent to input signals from 1# cutting tool to 8# cutting tool.

EROR is correspondent to input signal of XERR, YERR and ZERR (drive alarm).

STRT and PAUS is correspondent to external start and pause signals..

EMRG is correspondent to external emergency signal.

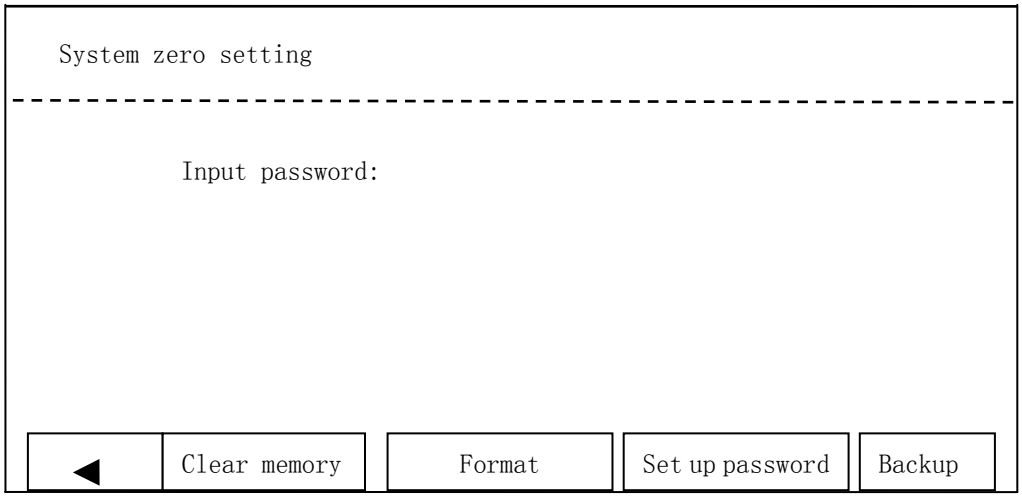
LIMT and LIM is correspondent to positive and negative limit signals.

SPHL is correspondent to spindle high/low speed input signals

4.10 System zero setting

Because the system has power down protection function, so it's impossible for system to initialize the storage area protected by power down after starting up for each time, else the parameter or program will be damaged. However, after the system is installed or user feels it's necessary to "Clean" this interval, "F5 (system zero setting)" can be pressed, and it sets all storage cells as 0 which includes all adaptable parameters, processing programs and program name tables. Accordingly, user should execute this function with caution for preventing losing important programs. Therefore, the system sets up one security measure, which only the operator with "Clean" system right can use this function:.

(1) Pressing "F5 (system zero setting)" to access system initialization, and the screen displays as follows:



In the screen, there is 'Input password:' and displays cursor..

(2) Inputting the password corresponding to this system. These characters aren't displayed in the screen to keep secret. . Finally pressing the enter key..

(3) If the input password is wrong, it will be noted by the system and waits for input again..

(4) After the input is right, the memorizer can be "Cleaned" with F2-F4..

4.10.1 Clear memory

After 4.10 operation is completed, pressing "F2 (clear memory)" to clear system data region and returning to initial state of starting up.. The operation of clearing memory should be used if the conditions below occur: The abnormal conditions during the system operation such as system down, system disorder owing to external interference, disorder of display and so on.

4.10.2 Format

After 4.10 operation is completed, pressing "F3 (Format)" to format electronic disk (clear user programs and all system parameters)..

When the user program is error, and disorder of file or file directory occurs, the electronic disk can be deleted through formatting. The user should implement this function with caution for preventing losing important programs.

4.10.3 Password setup

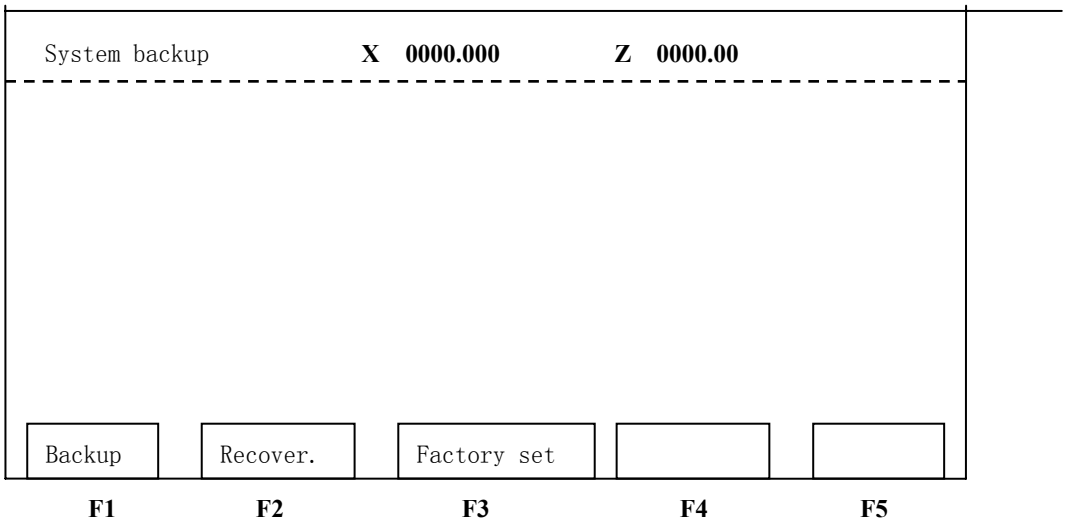
After 4.10 operation is completed, pressing "F4 (password setup)".. The password can be set by user himself, when pressing "F4" to change password, the new password should be input

first, then the system notes to input it again. If they are identical, the system will accept the new password. The system password of factory set is "XZ 0012"..

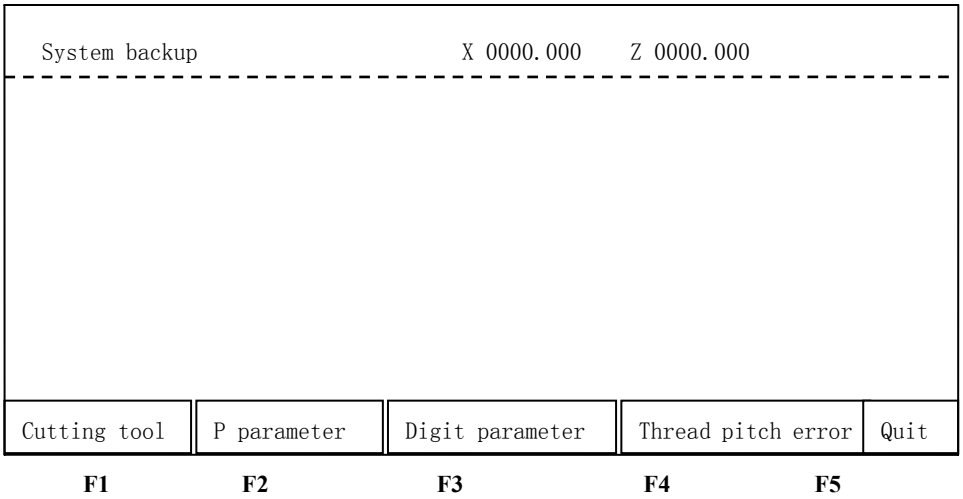
Then user can implement other operations again.. (various parameters required for machine tool operation should be input first). In order to prevent losing important programs and parameters, this password is prohibited for users without system "clear" right. This system can guaranty that every machine tool has different password when leaving factory, or makes it identical in the batch of systems of the same user offered according to user's requirements.

4.10.4 Backup

After 4.10 operation is completed, pressing "F5 (Backup)" to backup and recover the cutting tool parameter, P parameters, digit parameters and thread pitch error parameters, or set up them as defaults of factory set. The display is as follows:.



After pressing "F1 (Backup)" or "F2 (Recover)" or "F3 (Factory set)", the display is as follows:



Pressing F1-F4 can backup/recover/set up to factory set towards to corresponding parameters. After completing operation, 'Saved' or 'Recovered' are displayed in the lower right corner of the screen. The previous screen can be returned by pressing "F5 (Quit)".

4.11 Shutdown and starting up

Pressing "◄►" key in the PAKAM (Parameter) main image, and the screen displays:.

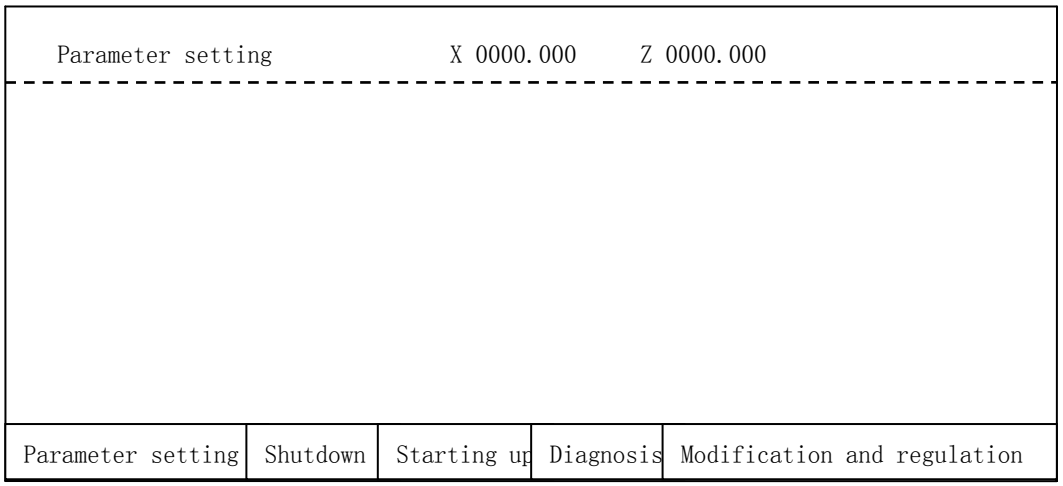


图 **3-6**

Pressing "F1 (Shutdown)": CNC memorizes some important parameters such as current coordinate, clearance direction, cutting tool compensation number and so on, which are saved or saved on the electronic disk, then shutdown can be made..

Pressing "F2 (Starting up)": CNC reads the parameters saved in the last shutdown, and recovers current coordinate, clearance direction, cutting tool compensation number and so on.. After operation is completed, 'Completion' is displayed in the top right corner of screen

Appendix One--Error alarm

Notes of error number and error content:

Error number	Error content
01	The time defined by G04 has error.
02	Undefined K parameter.
03	Error of G24 sub-program returning, and disorder of transition processing and sub-program call.
04	Data exceeds after G31 magnifies.
05	The parameter error of writing cutting tool.
06	The cutting tool number or cutting tool compensation number have errors when starting up.
07	Without the G and M functions.
08	Transition processing nestification has error (including sub-program call)
09	CNC doesn't memorize work piece coordinate before calculating cutting tool compensation value, using Xsav or Zsav keys.
10	The first letter of program line has error (the line must be initiated with N)
11	The arc parameter is not complete.
12	File directories have full.
13	The data format has errors such as numerical value after coordinate, the cutting tool compensation value which is correspondent to cutting tool number (such as T01) in the cutting tool parameter table, numerical value in the P parameter, four digits are required before decimal and three digits after decimal.
14	Transition processing doesn't find completion segment number
15	There are too many characters or illegal characters appear in the line
16	Undefined.
17	Transition processing nestification has error.
18	Parameter exceeds scope.
19	The radius of arc track is less than the cutting tool's.
20	There have been M02 and M30 when compiling M00.
21	What's called by G20 is not sub-program.
22	Data is too big
23	The thread pitch is too big or too small or thread lacks Z and K
24	The object segment number of cycle processing has error or is not found
25	Undefined.
26	Transition processing shouldn't appear in the last line, and should add M02.
27	The arc lacks parameter.

28	Undefined.
29	Undefined.
30	Undefined.
31	Undefined.
32	File is not found or file has error.
33	File has been damaged.
34	Memory is full.
35	Evolution data is too big.
36	Undefined.
37	Undefined.
38	Undefined
39	Two parameters after G and M have errors
40	Limit
41	Drive alarm.
42	General alarm.
43	Undefined.
44	The time is too long for holder positioning.
45	Definition of filename has error, or the communication has error when implementing input/output.
46	The memory of system is not enough to edit this file.
47	Undefined.
48	This file can't be used owing to disorder of the file internal addresses
49	The file directory is damaged.
50	The arc start point doesn't conform to end point.
51	Undefined.
53	When the optional segment starts, the corresponding segment number is not found.
55	Emergency stop alarm.
66	The cutting tool compensation number isn't checked.
71	There is key pressed when starting up
72	Thread is too short to reduce speed
73	The thread acceleration is wrong or the length of thread is too short to increase and decrease speed
74	The illegal processing data is found (CNC internal error)
76	The length of thread is too short to raise speed/measurement has error
80	Without this I/O port.
84	The file is read from electronic disk with error.
85	That too many data is sent from serial port causes insufficient memory.
86	The file of system parameter can't be read from electronic disk, please format electronic disk and recover the factory default value of four parameter tables.

Appendix Three Programming Example

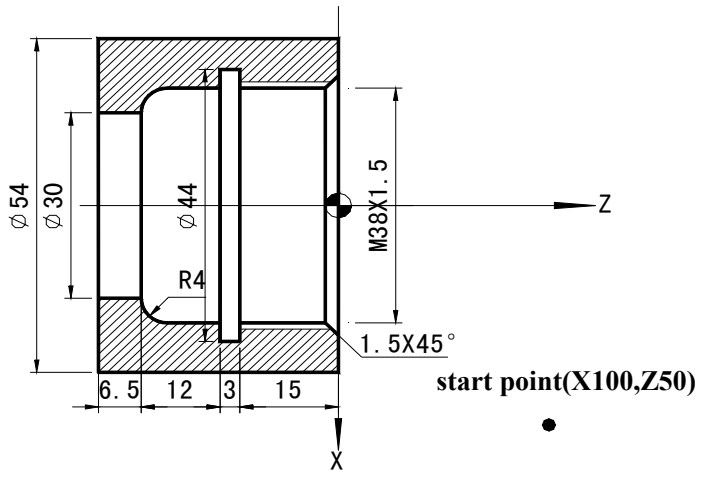
Example One The following elements are given with the figure

Material: Brass Rough: Forging, single side allowance is about 1mm

Thread is the metric system straight thread, and the thread pitch is 1.5 mm

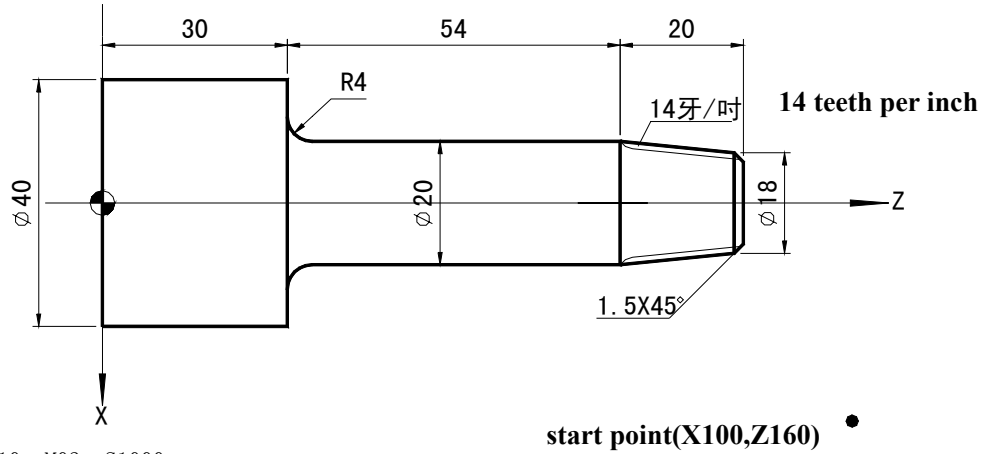
1# cutting tool: Inner bore cutting tool; 2# cutting tool: groved cutter (the groove width of cutting tool is 3mm); 3# cutting tool: Screwer

The start point of cutting tool is (X100, Z50);




```
N0010  M03  S1500
N0020  G00  X100  Z50
N0030  TI
N0040  G00  X30
N0050  G00  Z0
N0060  G01  X55  F150;   (Processing end surface)
N0070  G01  X39.5
N0080  G01  X36.5  Z-1.5;  (Chamfer 1.5X45)
N0090  G01  Z-26;  (turning inner bore diameter 38)
N0100  G03  X30  Z-30  R4  F100;   (turning inner arc R4)
N0110  G01  Z-37;   (turning inner bore diameter 30)
N0120  G00  X28  (X direction retracting)
N0130  G00  Z50  (Z direction retracting)
N0140  T2;   (changing inner bore tool to groove)
N0150  G00  X35;   (fast feed)
N0100  G00  Z-18;   (fast feed)
N0170  G01  X44  F150;   (grooving)
N0180  G00  X36;   (fast retracting)
N0190  G00  Z50;   (fast retracting)
N0200  T3;   (changing screwer to process thread)
N0210  S700
N0220  G00   X36.5  Z2;    (fast feed)
N0230  G86  Z-16  K1.5  I-4  R2  L4;   (processing thread)
N0240  G00  X100  Z80
N0250  M05
N0260  T1
N270  G00  X100  Z250
N0280  M02
```

Exemple Two. The following elements are given with the figure
Material: 45# Rough: Forging, single side allowance is about 1 mm,
Thread is the English system taper thread, and the thread pitch is 14 teeth per inch,
1# cutting tool: excircle cutting tool, 2# cutting tool: external thread cutting tool
The start point of cutting tool is (X100, Z160);



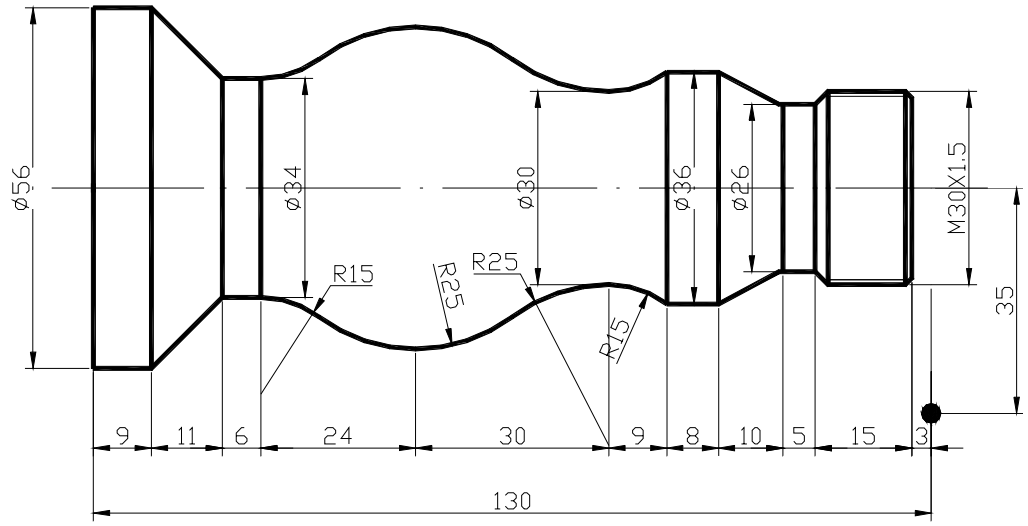

```
N0010  M03  S1000
N0020  M08
N0030  G00  X100  Z160
N0040  T1
N0050  G00  X44  Z30.2;(fast feed)
N0060  G01  X30  F120;   (rough turning end surface)
N0070  G00  Z107;   (fast retracting)
N0080  G00  X18.4;(fast feed)
N0090  G01  Z104  F120;   (low feed)
N0100  G01  X20.4  Z84;    (rough turning external cone, the diameter allowance is 0.4)
N0110  G01  Z34;   (rough turning excircle 20)
N0120  G02  X28  Z30.2  R3.8  F80;   (rough turning R4)
N0130  G01  X40.2  F120;   (rough turning end surface)
N0140  G01  Z0;   (rough turning excircle  40)
N0150  G00  Z104;   (fast retracting)
N0160  S1500
N0170  G00 X24;   (fast feed)
N0180  G01  X-0.2  F120;   (fine turning end surface)
N0190  G01  X15
N0200  C01  X18  Z102.5;   (chamfer 1.5x45)
N0210  C01  X20  Z84;   (fine turning taper)
N0220  C01  Z34;   (fine turning excircle diameter 20)
N0230  G02 X28  Z30  R4  F80;   (fine turning R4)
N0240  G01  X40  F120;   (fine turning end surface)
N0250  G01  Z0;   (fine turning excircle diameter 40)
N0260  G00  X100  Z160;   (fast retracting)
N0270  T2;   (changing screwer to process thread)
N0280  S700
N0290  C00 X17.8  Z106;(fast feed)
N0300  G87 X20  Z84  K14  I4  R1.96  L8;   (turning taper thread)
N0310  G00  X100  Z160
N0320  M05
N0330  M09
N0340  T1
N0350 C00  X100  Z160
N0360  M02
```

Example Three. The elements showed in the figure below have three cutting tools in total.

1# cutting tool: It's the 90 degree excircle, 2# cutting tool: 5 mm (grooved cutter), 3# cutting tool: 60 degree screwer

Rough dimension 60X150, Material: Aluminum.

The programming is as follows:




```
N0010  G00  X70  Z130
N0020  M03  S800
N0030  G01  Z127  FS0;   (low feed)
N0040  X-0.5;   (turning end surface)
N0050  G00  Z130;   (fast retracting)
N0060  X56.2;   (fast retracting)
N0070  C01  Z0  F80;   (rough turning excircle 56)
N0080  G00  X58;   (fast retracting)
N0090  Z130;   (fast retracting)
N0100  G01  X50.5  F80;   (low feed)
N0110  Z14;   (rough turning excircle)
N0120  G00  X52;   (fast retracting)
N0130  Z130;   (fast retracting)
N0140  G01  X44  F80;   (low feed)
N0150  Z70;   (rough turning excircle)
N0160  C00  X46;   (fast retracting)
N0170  Z130;   (fast retracting)
N0180  C01  X40  FS0;   (low feed)
N0190  Z70;   (rough turning excircle)
N0200  G00  X42;   (fast retracting)
N0210  Z130;   (fast retracting)
N0220  G01  X36.2  F80;   (low feed)
N0230  Z75;   (rough turning excircle)
N0240  G00  X38;   (fast retracting)
N020  Z130;   (fast retracting)
N0260 G01  X28.5    F80;   (low feed)
N0270  X30.5  Z125;   (chamfer)
N0280  Z104;   (rough turning excircle)
N0290  G00  X90;   (fast retracting)
N0300  Z200;   (fast retracting)
N0310  T02
```

```
N0320  M03  S400
N0330  G00  Z107;   (fast feed)
N0340  X32;   (fast feed)
N0350  G01  X26.2  F20
N0360  G00  X52;   (fast retracting)
N0370  Z20;   (fast retracting)
N0380  G01  X34.2  F20;   (rough turning excircle)
N0390  C01  X52  FS0;   (low retracting)
N0400  G00  Z200;   (fast retracting)
N0410  T3  S1200
N0420  G00  X32  Z127
N0430  C01  X30  FS0;   (low feed)
N0440  Z114;   (fine turning thread excircle diameter 30)
N0450  X26  Z112;   (chamfer)
N0460  Z107;   (fine turning excircle diameter 26)
N0470  X36  Z97;   (fine turning taper)
N0480  Z89;   (fine turning excircle 36)
N0490  X54  Z80;   (rough turning excircle)
N0500  X38  Z26;   (rough turning excircle)
N0510  Z20;   (rough turning excircle)
N0520  X58  Z9
N0530  G00  Z97;   (fast retracting)
N0540  G01  X36  F80;   (low feed)
N0550  Z89;   (fast retracting)
N0560  G02  X30  Z80  R15;   (fine turning clockwise arc R15)
N0570  G02  X40  Z65  R25;   (fine turning clockwise arc R25)
N0580  G03  X40  Z35  R25;   (fine turning reversion arcR25)
N0590  G02  X34  Z26  R15;   (fine turning clockwise arc R15)
N0600  G01  Z20;   (fine turning excircle diameter 34)
N0610  X56  Z9;   (fine turning taper)
N0620  Z0;   (fine turning excircle diameter 56)
N0630  S400
N0640  G00  X58;   (fast retracting)
N0650  Z132;   (fast retracting)
N0660  X30;   (fast feed)
N0670  G86  Z110  K1.5  I4  R1.35  L8;   (turning thread)
N0680  G0  X70;   (fast retracting)
N0690  Z200;   (fast retracting)
N0700  T1
N0710  M02
```